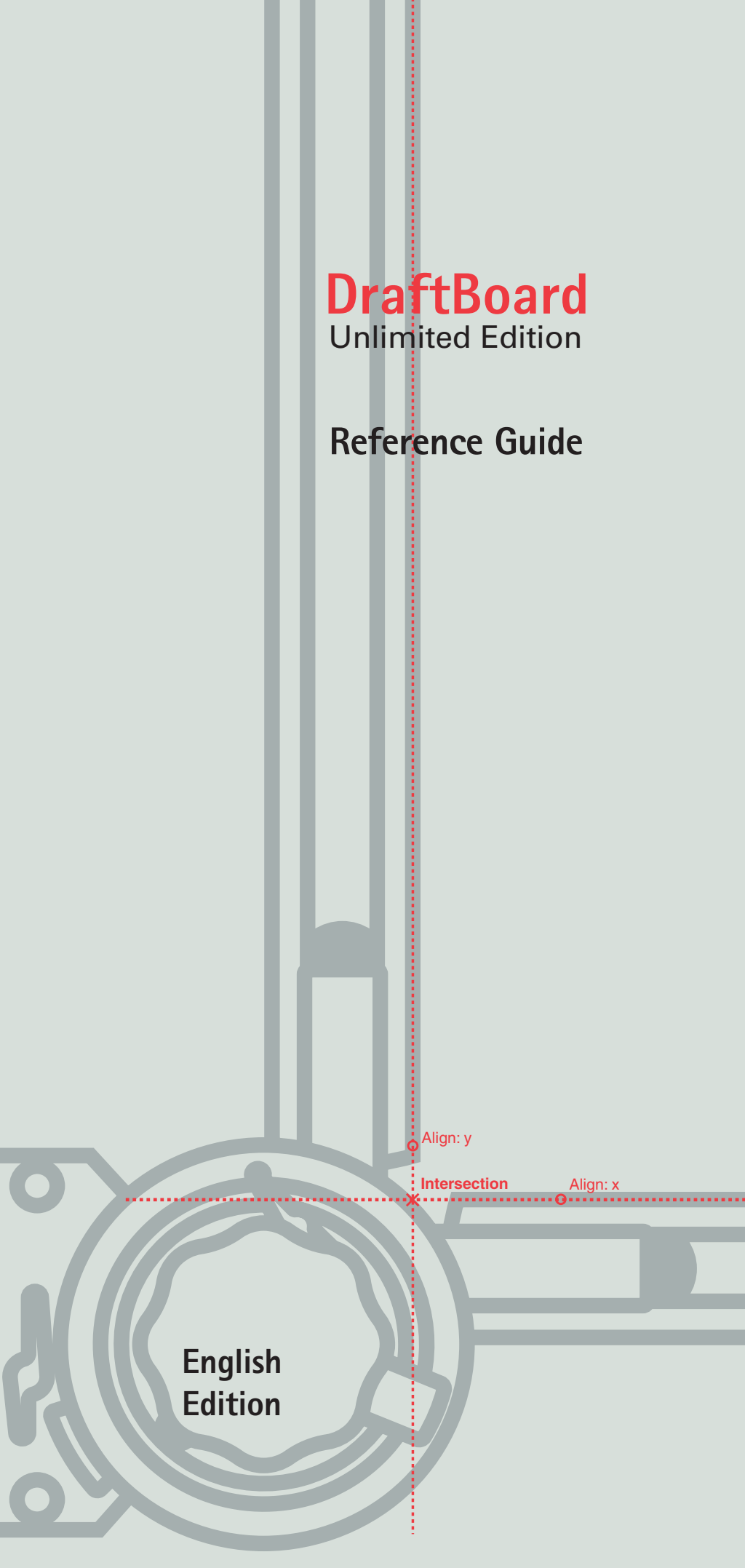
DraftBoard
Unlimited Edition

Reference Guide



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Unlimited Edition

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CHAPTER

1

Documentation

Documentation

The following manuals describe ***DraftBoard Unlimited*** for *Windows 98, Windows ME, Windows 2000, Windows NT4 SP5*, *Windows XP* and for the *Mac*.

Before using this manual, however, you will need to install ***DraftBoard Unlimited***. The installation is described in the *Getting Started* chapter of the *Tutorial*. After installation, we encourage you to continue with the tutorial exercises included in the Tutorial. This will familiarize you with the tools, features and commands of ***DraftBoard*** and enable you to maximize your productivity in the shortest amount of time.

Manual Layout

The ***DraftBoard*** documentation consists of the following:

- *Reference Guide*
- *Getting Started and Tutorials*

When the user interface differs only by its appearance not by its functionality the accompanying graphics alternately display the supported platforms.

In cases where ***DraftBoard*** functions differently on each specific computer platform instructions are provided for each platform.

When a function is available only for one of the supported platforms a note is displayed in the page margin.

WIN = **Windows**
MAC = **Macintosh**

Tips and advanced user notes are displayed in the margin throughout the manual. You don't need to know this information to run ***DraftBoard***. If you are a novice user, you may want to skip it entirely.

Reference Guide

This manual describes how to use ***DraftBoard*** including all its pertinent tools and commands. If you want information on a particular tool or command, simply look it up in the *Index*. Finally there is a *Task Index* which groups various tasks according to their functions.

Parametrics

This *Reference Guide* also includes an in-depth look at the integrated *Parametrics*, that may be a new feature to you. You can use ***DraftBoard*** to perform your daily tasks without understanding *Parametrics*, but if you do a lot of repetitive drawing or modifications of existing drawings, you will find this feature an invaluable time saver. It is particularly useful if you are a member of a design team that works on the same parts.

3D Modelling

The second part of this *Reference Guide* describes the 3D modelling process in ***DraftBoard Unlimited***.

Even ***DraftBoard*** offers a seamless transition from 2D to 3D, you should already know how to use the 2D tools and commands that are described in detail in the first part of this manual.

IMPORTANT! = Not all 3D Features (such as **AutoSurface**) are in all country versions and/or for all operating systems available. Check with your local distributor.

Appendices

At the end of this manual you find several appendices with additional information about specific settings for some ***DraftBoard Unlimited*** *components.*

Operators and Units

This section describes all *Operators* and *Units* which are accepted by all ***DraftBoard Unlimited entry*** boxes.

Special Characters

This section lists all the special characters and symbols you can use in ***DraftBoard Unlimited which*** are not directly available from the keyboard.

Date Interfaces

This appendix contains additional information about the *Data Interfaces* integrated into ***DraftBoard Unlimited.***

Plotter Fonts

This section lists all plotter fonts available in ***DraftBoard Unlimited*** that you can use in addition to the *PostScript* and *TrueType* fonts that are installed on your computer.

Hatch patterns

DraftBoard Unlimited contains a library with many ***ISO*** and ***DIN*** *hatch patterns*. This appendix lists all with ***DraftBoard Unlimited*** available hatch patterns and describes as how you can define your own hatch patterns and add them as ***.pat*** *files* to the ***DraftBoard Unlimited*** *hatch library*.

Registration

The *Customer Services* are described in the *Appendices* section. This appendix provides technical support, registration and update information. We'd really like to encourage you to return the registration card for ***DraftBoard Unlimited***, conveniently located inside your software box.

For such a minor investment of time, you'll become a registered customer. Only by registering ***DraftBoard Unlimited***, are you entitled to receive Telephone product support and Free updates.

Help

The integrated ***Online*** *Manual* provides a complete description of the program's many features, commands, and tools and a direct access to the Tutorial.

CHAPTER

2

Creating Geometry

Drawing Techniques

Drafting Aids

Drawing Tools

Creating Geometry

This chapter describes how to create the geometric objects that are the building blocks for your designs. It also describes how the *Drawing Assistant* helps you design faster and easier. The following topics are contained in this chapter:

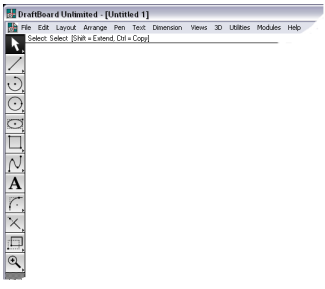
- Drawing Techniques
- Drafting Aids
- Drawing Tools

This chapter requires that you know how to *open* or *save* a file or how to use a *Zoom-tool*. If you are unfamiliar with *Windows* programs see the *Getting Started* section in the *Tutorial*.

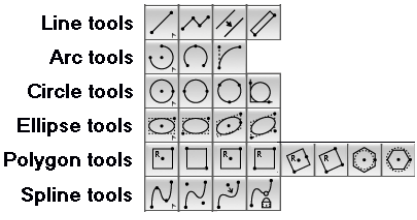
You will find more about the functionality of the **Drawing Assistant** in *Chapter **3***.

Function Palette

Drawing begins with the Tool palette, located at the left of the ***DraftBoard*** drawing area shown below:


DraftBoard Unlimited - [Untitled 1]
File Edit Layout Arrange Pen Text Dimension Views 3D Utilities Modules Help
Select: Select [Shift = Extend, Ctrl = Copy]

The Tool palette has six subpalettes of tools that you can use to create geometric objects. These six tools are the drawing tools.



Drawing Techniques

Most drawing tools allow two methods of creating an object: clicking and ***dragging***.

Clicking This method allows you to create objects by clicking points with the mouse. The benefit of this technique is that you can perform unrelated activities, such as zooming, while you are creating the geometry.

Dragging The ***drag*** method allows you to indicate two points by ***dragging*** from one to the other. Press and hold down the mouse button to set the first point. Next, ***drag*** the mouse to the location for the next point and release the mouse button. The benefit of this technique is that a *rubberband* image appears during the construction to show you how the object will appear once you release the mouse button.

You can use either or both methods to construct most objects.

To help make object creation easier, ***DraftBoard's*** tools contain a feature we call *Smart Pointers*. As you work with a tool, the tool icons show you the points you must indicate to create an object with the tool.

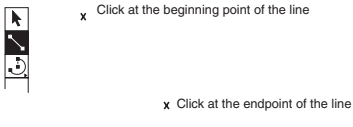
Each dot on the icon represents a point you must place either by clicking or by ***dragging***. In addition, the *Smart Pointer* indicates the order for designating points as you are drawing the geometry.

Creating a line by clicking

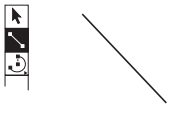
1. Select the **Single Line** tool from the Tool palette.

2. Move the pointer to the drawing area and click to set the starting point of the line.
3. Move the pointer to the desired location for the endpoint of the line and click.

If you use the clicking method to draw a line no *rubberband* line guides you.



The line is drawn between the two points.



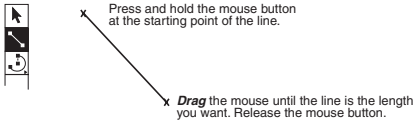
Immediately after construction, you can make changes in the Status Line to alter the length or position of the line. If the geometry isn't satisfactory, just press the **Backspace** key.

Creating a line by *dragging*

1. Click the **Single Line** tool in the Tool palette.

2. Move the pointer into the drawing area.
3. Press and hold down the mouse button at the desired location for the starting point of the line.
4. With the mouse button still held down, move the mouse to the desired location for the endpoint of the line.

Notice that when you ***drag*** between points, a *rubberband* line guides you. If you use the clicking method, you can perform unrelated activities, such as zooming, between clicks.


Press and hold the mouse button at the starting point of the line.
Drag the mouse until the line is the length you want. Release the mouse button.

5. Release the mouse button to set the endpoint.
   Immediately after construction, you can make changes in the *Status Line* to alter the length or position of the line. If the geometry is not satisfactory, just press the **Backspace** key.

Drafting Aids

For creating geometry objects the ***DraftBoard*** user interface provides the following drafting aids:

- Drawing area
- Message Line
- Smart Mouse Pointer
- Status Line
- Drawing Tools

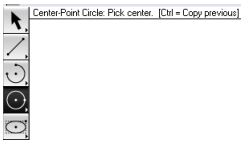
Drawing Area

All objects are created, modified and annotated on the drawing area. You can imagine this drawing area as an infinite large sheet of paper on which you can create any type of geometry object at its original size. Using the Scroll bars on the left and lower side of the drawing window you can move the sheet up and down or right and left. You can display different parts of the drawing sheet by ***dragging*** the slider of a scroll bar to the approximate location.

Message Line

The Message Line across the top of the drawing area provides concise instructions for the use of the current tool.

For example, after selecting the **Center-Point Circle** tool, the Message Line appears as illustrated below:



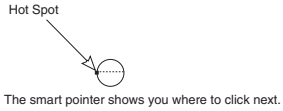
The instructions in the Message Line for some tools also indicate optional activities. For example, if you hold down the **Control** (Windows) or **Option** (Macintosh) key while using the **Center-Point Circle** tool, the next mouse click creates a copy of the last circle with the center placed where you clicked.

Smart Pointer

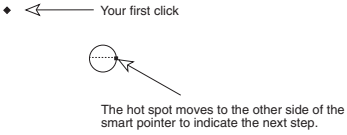
When you select a tool and move the pointer into the drawing area, the pointer shape is representative of the tool.

Some of the pointers, like the single line pointer, are simple cross-hairs. Others, such as the **Opposite-Point Circle** pointer, resemble the tool itself.

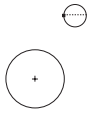
The pointer, called a *smart pointer,* displays indicators for multi-step procedures. Each smart pointer has a dot, the *hot spot*, showing the next point you should specify. The dot changes position on the pointer during each step of the construction.



For example, the **Opposite-Point Circle** pointer illustrated above shows that the first click of the mouse places a point on one edge of the circle you're creating. After you click a location, the hot spot moves to the other side of the pointer, showing that the next click places a point on the opposite edge of the circle.



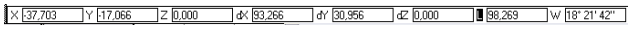
After you click the second location, the circle appears. The hot spot moves back to its original position on the pointer so that you can create another circle.



The Status Line

Whenever you select a tool from the Tool palette, the *Status Line* appears along the bottom of the drawing area. It contains boxes giving information about the current construction. For example, when the **Single** or **Connected Line** tool is the current tool, the *Status Line* contains boxes for the *x, y* and *z coordinates* of the beginning point and the *x, y* and *z values* for the endpoint of the line. It also contains the value of the *length* and *angle* of the line.

You can set the **number of decimal places** or fractional round off with the precision entry in the **Units** dialog box from **Preferences** in the **Layout** menu.



You can use the Status Line in three ways:

- To create an object with keyboard entries only.
- To edit an object that was just created and is still selected.
- To create an additional object using the current tool.

Whenever you construct an object, the status box containing the specification that you are most likely to change is selected (the box is highlighted).

For example, when you draw a line with the **Single Line** tool, the **Length** box is highlighted so you can enter a value for the length. Just type a value. When you press **Enter** the line is redrawn at the new length.

Using the Status Line

You can enter up to 10 characters in each **Entry field** of the Status line. Notice how each method affects the selection of the contents of the status box. Clicking once inside the box places the cursor where you clicked. If you double-click inside the box, the entire entry is selected so you can change it.

If you want to make an entry in a different status box, you can use one of four selection methods:

- Press the **Tab** key to cycle the selection highlight through the status boxes from left to right.
- Click inside the status box and the pointer becomes an **I-beam** text cursor.
- Click the box label to select the entire contents of the box.
- Double-click inside the status box to select the entire contents of the box.

Using the Status Line with Drawing Tools

Once you have drawn an object, you can adjust its specifications such as *length*, *angle*, and *location* in the status Line. You must make these changes immediately, before you construct another object, select a different tool, or choose a command. You can make only one series of changes in the status line (in as many boxes as necessary); after you press the **Enter** key ***DraftBoard*** redraws the object to your specifications. Once you have pressed the **Enter** key, you must make subsequent changes with the **Edit Objects** command in the **Edit** menu.

Try the following exercises to create and change a single line with the Status Line.

Altering geometry in progress with status boxes

1. Select the **Single Line** tool.
2. Click **2** locations in the drawing area.
   The length (**L**) box is highlighted in the status line.
3. Type **3**.
   The *3* is entered directly in the **L** box.
4. Press the **Tab** key to select the next status box.
   The angle (**A**) box is now highlighted.
5. Enter **15**.
6. Press the **Enter** key.
   Pressing the **Enter** key completes the data entry for this object. The line is redrawn **3** units long and at a **15°** angle.

Creating additional geometry with status boxes

1. With the **Single Line** tool still selected from the previous example, click **2** more points.
2. Type **4**.
3. Press the **Tab** key and type **25**.
4. Press **Enter**.

   Another line is drawn.

You must remember that when you press the **Enter** key, ***DraftBoard*** constructs the object based on the specifications in the status boxes. Pressing **Enter** a second time in the above example would create a second line with the same values. Since the lines have the same values, the second line overlays the first line and you can't see it.

Creating new geometry with status boxes

1. Select the **Selection** tool so the **Single Line** status box clears.
2. Click the **Single Line** tool again.

   The **X** status box is active, showing that you can enter a value for the X coordinate of the beginning point of the line.
3. Type **0**.

   Note: Do not press **Enter** until you reach step **#10**.
4. Press the **Tab** key.

   The **Y** box is highlighted.
5. Type **0**.
6. Press the **Tab** key.

   The **dX** box is highlighted. The dX value is the delta X, the numeric difference between the beginning and ending X coordinate.
7. Type **2**.
8. Press the **Tab** key.

   The **dY** box is highlighted.
9. Type **2**.
10. Press the **Enter** key.

    The line is drawn.

The *angle* and the *length* of the line are calculated by the values you have specified. If you specify values for the *angle* and the *length* of the line you don't have to enter values in the **dx** and **dy** status fields and vice versa.

Creating Geometry Offset from a Point

If you want to create some geometry that is offset from an existing point, you can use the *Status Line* to specify the offset.

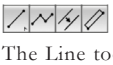
Creating geometry offset from a point

1. Select the tool you want to use.
2. Move the pointer over the *control point* from which you want the offset.
3. Click once to lock onto that point.
4. Click in the appropriate **X** or **Y** box in the Status Line, placing the text cursor at the end of the entry.
5. Type the offset (such as **+ 3"**) and press **Enter**.
6. Continue with your construction.

Drawing Tools

The following section describes the drawing tools found on the *Tool palette*.

Line Tools



The Line tools on the Tool palette create *line segments*, *connected lines*, *lines parallel* to existing lines, and *smart walls*. As you create a line, the coordinate locations, line length, and angle from horizontal appear in the Status Line.

The line is drawn with the current pen specifications for *color*, *weight*, and *pattern*.

Single Line Tool

This tool draws a line between two points. You can click or ***drag*** to draw a line.

Using the Single Line tool

- Click two endpoints of the line.

  Or:

- ***Drag*** to indicate the endpoints of the line; press at the beginning and release at the end of the line. As you ***drag***, you see a *rubber-band* line that previews your construction.

Notice that after completing a line the Message Line says **Ctrl = Copy** (*Macintosh:* **Option=copy**). This indicates that if you hold down the **Ctrl**-key (*Macintosh:* ***Option*** *key*) and click once in the drawing area, a line will appear that is identical to the one just drawn. The location of the click designates the location of the first point.

The Status Line allows you to specify the **X, Y** *coordinates* of the beginning, the relative location of the end (*delta X* and *delta Y*), the line *length*, and the *angle* from horizontal. Once a line is drawn, the line **Length** is the selected status box.

Drawing a line perpendicular to another object

1. Construct the line (or spline).
2. Move the pointer to the object until a Drawing Assistant ***on*** notation appears.
3. ***Drag*** straight away from the object in a ***perpendicular*** direction.
4. ***Drag*** the extent of the line.

Drawing a line tangent to or perpendicular to a curve

1. Construct an arc, circle, or ellipse.
2. Choose the **Single Line** tool.
3. Move the pointer to the arc until a Drawing Assistant ***on*** notation appears. The Drawing Assistant notation must be ***on*** rather than ***endpoint***, ***quadrant***, or ***midpoint***.
4. ***Drag*** in the appropriate direction (straight out for perpendicular or at an angle for tangent) until the Drawing Assistant ***perpendicular*** or ***tangent*** notation appears.
5. When the Drawing Assistant locks on to ***perpendicular*** or ***tangent***, you can then ***drag*** the line around the arc to the location you want and extend the line to the length you want.

Creating a point

You can create a point by creating a line and setting its length to zero. Set the starting point of the line with the mouse, then type **0** in the **Length** entry field of the status line and press the **Enter** key. The line ("point") displays a **+** to indicate its location.

When creating a point with the **Single line tool** by clicking twice the same position you have to select a different tool before you continue to draw. Otherwise you would create a line with a double *Starting point*. This is not required when you use the **Circle tool** to create a point.

You can also double click the **Single Line**, the **Center-Point Circle** or the **Opposite-Point Circle** tool in the same spot to create a point. These points can be selected and deleted as any other point. These points that are displayed by a **+** on the screen are printed as points.

Connected Lines Tool

This tool draws lines in which the endpoint of one line segment is the beginning point of the next.

Using the Connected Lines tool

1. Click or ***drag*** to indicate the endpoints of the line segments.

   If you click a point and then change your mind, press the **Escape** key to remove the last line. Pressing the **Delete** or **Backspace** key or choosing **Undo** removes all connected lines in the current construction.
2. Indicate the last point by double-clicking or by choosing another tool.

The radius of a arc created with the **Connected line tool** can be modified with **Edit objects** command.

If you want to create a connecting arc between two points rather than a line, hold down the **Ctrl** key (*Macintosh:* ***Option*** *key) and* then click the endpoint of the segment. You must have constructed one line segment before you can use the arc function.

The status line allows you to specify the *X*, *Y* and *Z coordinates* of the beginning, the relative location of the end (delta X and delta Y), the line *length* and the *angle* from horizontal. Once the line segment is created, line **Length** is the selected status box.

Parallel Lines Tool

This tool constructs lines parallel to existing lines. Use one of the following methods for creating parallel lines:

Using the Parallel Lines tool

- Select the **Parallel Lines** tool, ***drag*** the line; press to select the line, and release to indicate the position for the parallel line. A rubberband line moves with the pointer.

  Or

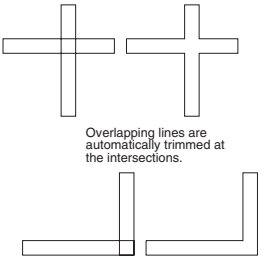
- Select the **Parallel Lines** tool, click the line you want to duplicate, and then enter a delta distance in the status line and press **Enter**. (This method does not allow you to control on which side of the original line the new line is drawn.)

The second method allows you to control the direction of the offset by entering **+** or **-** in front of the indicated values. Entering **+** offsets the object to the *right* or *downwards* on the screen, entering **-** to the *left* or *upwards*.

The status box allows you to specify the **distance** between the selected line and the parallel line.

Smart Wall Tool

This tool works in much the same way as the **Single Line** tool except that it draws double lines or **walls**. Walls created on the same layer are automatically trimmed to where they touch or intersect.



Automatic trimming of smart walls occurs only when the walls are on the same layer.

As you ***drag*** the mouse, the defining side of the wall falls on the construction line you drag along. The other side of the wall automatically fills in at the thickness you specify in the status line. The position of the automatic wall depends on your last activity. If the second wall is on the wrong side of the construction line, press the **Shift** key to flip it.

Using the Smart Wall tool

1. Select the **Smart Wall** tool.
2. In the status line, specify the thickness (**T**) of the wall.
3. Click or ***drag*** to indicate the ends of the wall segment.

   You can press the **Shift** key to flip the wall to the opposite side.
4. Continue specifying both endpoints for wall segments, as needed.

If you delete a merged segment, the remaining walls redraw.

If you want to *fillet* or *chamfer* smart walls you have to ungroup them with the **Ungroup** command in the **Arrange** menu.
It is recommended that you place all symbols, before you *ungroup* and *fillet/chamfer* the walls, since ungrouped walls loose their smart features such as automatic trimming. Select only the wall segments not any placed symbols if you want to ungroup a wall.

The status line allows you to specify thickness of the wall, the **X**, **Y**, **Z** coordinates of the beginning point, the wall length and the angle from horizontal. Once the wall segment is created, wall **Length** is the selected status box.

Wall symbols

Parametrized symbols with an underlying wall segment like the door symbols you find in the *Architect Library* will automatically cut wall segments where they are placed. When you move the symbol along the

Parametrics works with some but not all wall configurations. If you use parametrics, turn on the point display (**Show Points** on the **Layout** menu) and dimension to the displayed points rather than to the lines. Keep in mind that double walls obscure the fact that smart walls are actually single lines.

wall to another position the wall will automatically be closed at the old position and be cut open at the new position.

If you want to use the **Smart Wall** feature for single lines, you have to create the lines with the **Smart Wall** tool and enter **0** for the thickness of the wall.

Arc Tools

You can use the Arc tools on the tool palette to create an arc by any of three methods:

- *Center-Point Arc*
- *3-Point Arc*
- *Tangent Point Arc*

As you create each arc, the status line displays entries such as coordinate locations, radius, angle from horizontal, and delta angle.

The arc is drawn with the current pen specifications for *color*, *weight*, and *pattern*.

Center-Point Arc Tool

This tool draws an arc from three points: the center point, arc beginning point, and arc endpoint.

Using the Center-Point Arc tool

1. Click the ***center*** point of the arc.
2. Click the radius and beginning of the arc.
3. ***Drag*** the extent of the arc, pressing at the beginning point of the arc and releasing at the endpoint.

You can simply click all three points, but the *rubberband* arc doesn't appear when you use this method.

To construct an arc greater than **180°**, you must ***drag***, rather than click, the endpoint.

The status line allows you to specify the *X, Y coordinates* of the center of the arc, the *length* of the radius, the starting *angle* from horizontal, and the delta angle from the start. **Radius** is the selected status box.

3-Point Arc Tool

This tool draws an arc through the points you select.

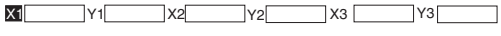
Using the 3-Point Arc tool

1. Click the first position on the arc.
2. ***Drag*** or click the second and third positions. (***Dragging*** displays the *rubberband* arc.)

If ***DraftBoard*** can't create the arc tangent to an object it will define an alternate construction.

The arc is drawn from the first position that you indicate in the direction of the next point.

If you click ***on*** an object while holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)*, the arc is drawn tangent to the specified object rather than through the point. You may combine the placement of points to create an arc through a specific point of one object and tangent to another object, or an arc tangent to three objects, or any other combination.



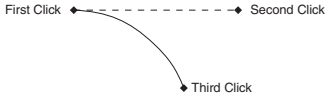
The status line indicates the X, Y *coordinates* for each of the three points.

Tangent-Point Arc Tool

This tool draws an arc beginning at the first point you specify. The second point you specify is the direction vector, and the third point you specify indicates the endpoint of the arc. Essentially, the **Tangent-Point Arc** tool first creates a line, then creates an arc tangent to the line, then erases the line.

Using the Tangent-Point Arc tool

1. Click the starting point.
2. Click the tangent vector. (The arc is tangent to a line drawn from the starting point to this point.)
3. Click the endpoint. The arc is drawn between the first and last point you click and tangent to the line between the first and second points.



The status line allows you to specify the X, Y *coordinates* of the endpoints of the arc and the *angle* of the tangent line.

Circle Tools

The Circle tools on the tool palette construct circles by any of four methods:

- *Center-Point Circle*
- *Opposite-Point Circle*
- *3-Point Circle*
- *Tangent-Point Circle*

Center-Point uses the center and diameter of the circle. *Opposite-Point* uses the diameter. **3-Point** uses three points or can be tangent to existing objects. *Tangent-Point Circle* draws a circle tangent to two objects, using the diameter.

The circle is drawn with the current pen specifications for *color*, *weight*, and *pattern*.

If you want to ***drag*** a circle by its center later on, you must use the **Move tool**, since the **Selection tool** doesn't display the center of the circle when selecting it.

Center-Point Circle Tool

This tool draws a circle specified by the center point and diameter.

Using the Center-Point Circle tool

- Click two locations; the first click places the center and the second determines the radius, or
- ***Drag***, pressing at the center and releasing at a position indicating the radius.

If you ***drag*** the points, a *rubberband* image guides your construction. You can create a copy of the last circle by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the center for the new circle.

The status line shows the X, Y *coordinates* of the center and the *diameter* of the circle. **Diameter** is the selected status box.

Opposite-Point Circle Tool

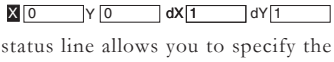
This tool draws a circle specified by the diameter.

Using the Opposite-Point Circle tool

- Click two locations to indicate the diameter.

  Or:
- ***Drag***, pressing the mouse button at the beginning and releasing it at a position that indicates the diameter.

If you ***drag*** the points, a *rubberband* image guides your construction.



The status line allows you to specify the **X**, **Y** *coordinates* representing the endpoints of the diameter.

3-Point Circle

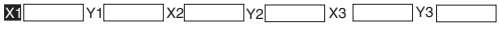
This tool draws a circle through the points you select.

Using the 3-Point Circle tool

1. Click the first point on the circle.
2. ***Drag*** or click the second and third points. (***Dragging*** displays the *rubberband* circle.)

If you place any of the three points on an existing object, the circle is drawn through that point. If you click an object while holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)*, the circle is drawn tangent to the object rather than through the indicated point. You may combine the placement of these points to create a circle through a specific point of one object and tangent to another object, or a circle tangent to three objects, or any other combination.

If you ***drag*** the first two points, a *rubberband* image guides your construction.



The status line indicates the X, Y *coordinates* for each of the three points.

Tangent-Point Circle Tool

This tool draws a circle tangent to the two objects you indicate.

Using the Tangent-Point Circle tool

1. Click the objects to which the circle is to be tangent.
2. You can enter a diameter for the circle in the status box.

Ellipse Tools

These tools on the tool palette construct ellipses inscribed within an invisible rectangle or parallelogram.

- The *2-Point Center* ***Ellipse*** uses the center point and one corner location of the rectangle.
- The *Opposite-Corner* ***Ellipse*** uses opposite corners of a rectangle.
- The *3-Point Center* ***Ellipse*** uses the center point, the midpoint of a side and the corner of the parallelogram.
- The *3-Corner* ***Ellipse*** uses three corners on a parallelogram.

You can click or ***drag*** to create the ellipse. ***Dragging*** creates a rubberband ellipse, so you can see the ellipse before it is drawn.

The ellipse is drawn with the current pen specifications for *color*, *weight*, and *pattern*.

Ellipses do not have ***center*** points by design. If you draw lines between opposite ***vertex*** points on the ellipse, then the ***intersection*** of the two lines will be the center of the ellipse. For a ***2 point Center*** *Ellipse* (inscribed in a rectangle) the Drawing Assistant's horizontal and vertical temporary construction lines will cross at the center point once you have touched two ***vertex*** points.

2-Point Center Ellipse Tool

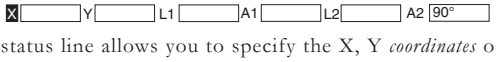
This tool constructs an ellipse inscribed within the rectangle calculated from the two specified points: the center point and one corner of the rectangle.

Using the 2-Point Center Ellipse tool

1. Indicate the center of the ellipse.
2. Indicate the corner of the rectangle defining the ellipse.

If the two points are on the vertical or horizontal axis, a straight line is drawn.

You can create a copy of the last ellipse by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the center.



The status line allows you to specify the X, Y *coordinates* of the center point and the *length* and *angle* of the semimajor and semiminor axes of the ellipse.

Opposite-Corner Ellipse Tool

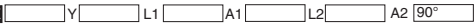
This tool draws an ellipse inscribed in the rectangle specified by the opposite corners you indicate.

Using the Opposite-Corner Ellipse tool

1. Indicate one corner of the rectangle defining the ellipse.
2. Indicate the opposite corner of the defining rectangle.

If the two points are on the vertical or horizontal axis, a straight line is drawn.

You can create a copy of the last ellipse by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the lower-left point.



The status line allows you to specify the X, Y *coordinates* of the lower-left point and the *length* and *angle* of the major and minor axes of the ellipse.

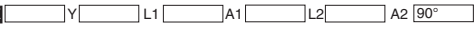
3-Point Center Ellipse Tool

This tool constructs an ellipse inscribed within the parallelogram calculated from three specified points: the center point, the midpoint of a side, and a corner of the parallelogram.

Using the 3-Point Center Ellipse tool

1. Indicate the center of the ellipse.
2. Indicate the midpoint of the side of the rectangle defining the ellipse.
3. Indicate the corner of the rectangle defining the ellipse.

If the three points are on the vertical or horizontal axis, a straight line is drawn. You can create a copy of the last ellipse by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the center.



The status line allows you to specify the X, Y *coordinates* of the center point and the *length* and *angle* of the sides of the parallelogram.

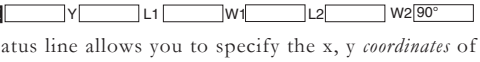
3-Corner Ellipse Tool

This tool draws an ellipse inscribed in the parallelogram calculated from the three corners you specify.

Using the 3-Corner Ellipse tool

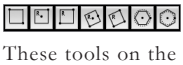
1. Indicate one corner of the parallelogram defining the ellipse.
2. Indicate another corner of the defining parallelogram.
3. Indicate the final corner of the defining parallelogram.

If the three points are on the vertical or horizontal axis, a straight line is drawn. You can create a copy of the last ellipse by holding down the **Ctrl** key (Macintosh*:* ***Option*** *key)* and clicking where you want the lower-left corner of the parallelogram.



The status line allows you to specify the x, y *coordinates* of a corner and the *length* and *angle* of the sides of the parallelogram.

Polygon Tools



These tools on the tool palette draw rectangles, inscribed polygons or circumscribed polygons. Some of the rectangle tools create single lines which are individual objects or polygonal rectangles that can be selected and edited as a single object. ***Dragging*** to indicate the points of the polygon creates a rubberband image.

The polygon is drawn with the current pen specifications for *color*, *weight*, and *pattern*.

Use the entry boxes **D** and **Sides** in the status line to change any polygon when first creating it. After that, it becomes a series of separate line objects.

Rectangle Single Lines Tool

This tool draws a horizontal or vertical rectangle, using the opposite corners you specify.

Using the Rectangle tool

- Click the opposite corners of the rectangle.

  Or:

- ***Drag*** the opposite corners of the rectangle.

You can create a square by aligning the second point on the **45°** construction line. If the two points are on the vertical or horizontal axis, a straight line is drawn.

You cannot change the **Width** and the **Height** of a single lines rectangle in **Edit Objects**. You must edit each line individually, because rectangles become 4 line objects after you deselect the rectangle.

You can create a copy of the last rectangle by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the upper-left corner.

A default **1-inch** rectangle is drawn if you have not previously drawn a rectangle.

The status line allows you to specify the X, Y coordinates of the first point, as well as the width and height of the rectangle. **Width** is the selected status box.

If you want to draw a square from the center rather than opposite corners, use one of the other polygon tools, specifying four sides. You can't create a rectangle from the center.

Rectangle *Corner-Center*

This tool draws a horizontal or vertical rectangle as a single object, using the *center* point and one *corner* point of the rectangle.

Using the Rectangle *Corner-Center*

- Click the *center* point and one *corner* point of the rectangle.

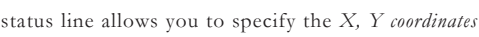
  Or:

- Click the center point and ***drag*** to the *corner* point of the rectangle (***Dragging*** displays a rubber band).

You can create a square by aligning the second point on the ***45°*** *construction* line.

You can create a copy of the last rectangle by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the *center* point.

A default **1-inch** square is drawn if you have not previously drawn a rectangle.



The status line allows you to specify the *X, Y coordinates* of the *center* point, as well as the *angle*, *length* and *height* of the rectangle. **Length** is the selected status box.

If you modify the *Angle* in the status line, the rectangle is rotated around its center point when you press the **Enter** key.

Rectangle *Opposite Corners*

This tool draws a horizontal or vertical rectangle as a single object, using the *opposite corner* points of the rectangle.

Using the Rectangle *Opposite Corners*

- Click the two *opposite corner* points of the rectangle.

  Or:

- Click one *corner* point of the rectangle and ***drag*** to the opposite *corner* point (***Dragging*** displays a rubber band).

You can create a square by aligning the second point on the ***45°*** *construction* line.

You can create a copy of the last rectangle by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the *center* point.

A default **1-inch** square is drawn if you have not previously drawn a rectangle.

X Y Angle Length Width

The status line allows you to specify the *X, Y coordinates* of the first *corner* point, as well as the *angle*, *length* and *height* of the rectangle. **Length** is the selected status box.

If you modify the *Angle* in the status line, the rectangle is rotated around its center point when you press the **Enter** key.

Rectangle *Center-Midpoint-Corner*

This tool draws a rectangle as a single object, using the *center* point, one *midpoint* and one *corner* point of the rectangle.

Using the Rectangle *Center-Midpoint-Corner*

- Click the *center* point, one *midpoint* for one side of the rectangle and one *corner* point of the rectangle (specifying the distance to the before defined axis).

  Or:

- ***Drag*** the mouse from the *center point* of the rectangle to one *midpoint* and click then the *corner* point of the rectangle.

The third point defines the distance from the axis defined by the ***centerpoint*** and the ***midpoint***.

You can create a copy of the last rectangle by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the *center* point.

A default **1-inch** square is drawn if you have not previously drawn a rectangle.

X Y Angle Length Width

The status line allows you to specify the *X, Y coordinates* of the *center* point as well as the *angle*, *length* and *height* of the rectangle. **Length** is the selected status box.

If you modify the *Angle* in the status line, the rectangle is rotated around its center point when you press the **Enter** key.

Rectangle *3 Corners*

This tool draws a rectangle as a single object specified by the 3 corners of the rectangle you indicate.

Using the Rectangle *3 Corners*

- Indicate 3 corners of the rectangle.

  Or:

- ***Drag*** the mouse from the *first corner* to the *second corner* and click then the *third corner* (specifying the distance to the before defined axis).

The third point defines the distance from the axis defined by the two ***corner*** points.

You can create a copy of the last rectangle by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the first corner of the rectangle.

A default **1-inch** square is drawn if you have not previously drawn a rectangle.

X Y Angle Length Width

The status line allows you to specify the *X, Y coordinates* of the first *corner* as well as the *angle*, *length* and *height* of the rectangle. **Length** is the selected status box.

If you modify the *Angle* in the status line, the rectangle is rotated around its center point when you press the **Enter** key.

Inscribed Polygon Tool

This tool draws a regular polygon where the radius of the circumscribing circle determines the location of the polygon's vertices. The default polygon is a hexagon, but you can specify the number of sides in the status line. (**Note**: The status line shows a diameter for the circle, the standard way of describing a polygon inscribed in a circle.)

Using the Inscribed Polygon tool

- Click the center of the polygon and a point on the circumference of the circumscribing circle.

  Or

- ***Drag*** from the center of the polygon to a point on the circumference of the circumscribing circle.

You can create a copy of the last inscribed polygon by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the center.

The status line allows you to specify the X, Y *coordinates* of the center, the *diameter* of the circle defining the polygon, and the number of *sides*. **Diameter** is the default status line selection, and the default number of sides is six.

Circumscribed Polygon Tool

This tool draws a regular polygon for which the radius of the circle determines the midpoint of the sides. The default shape is a hexagon, but you can specify the number of sides in the status line. (**Note:** The status line shows a diameter for the circle, the standard way of describing a polygon circumscribed around a circle.)

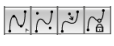
Using the Circumscribed Polygon tool

- Click the center of the polygon and the midpoint of one of the sides, or
- ***Drag*** from the ***center*** of the polygon to the ***midpoint*** of one of the sides.

You can create a copy of the last circumscribed polygon by holding down the **Ctrl** key *(Macintosh:* ***Option*** *key)* and clicking where you want the center.

The status line allows you to specify the X, Y *coordinates* of the center, the *diameter* of the circle defining the polygon, and the number of *sides*. The default number of sides is **6**. **Diameter** is the default status line selection.

Spline Tools



The spline tools on the tool palette create **NURBs** (**N**on-**U**niform **R**ational **B**-Splines) which are a superset of Bezier curves. These splines are curves created by a complex mathematical formula.

NURB splines provide designers with two interrelated functions. First, curvature continuity remains intact when the curve is changed. Kinks don't develop as the spline is altered. Second, NURB splines provide localized control of a complex curve. You can isolate an area and make changes without affecting the remainder of the spline.

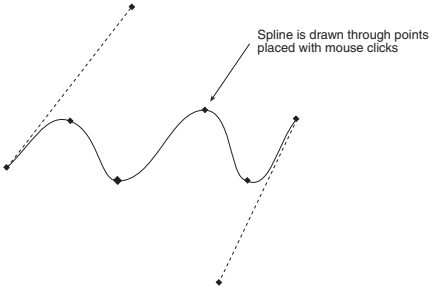
These properties are essential in aerodynamic designs. Air molecules moving over a wing surface must flow smoothly for maximum aerodynamic lift. If the surface does not maintain curvature continuity, the air molecules separate from the wing surface and cause a vacuum. Such a vacuum causes an eddy as the molecules try to fill it. This disruption of air flow increases the drag, which is not a part of an effective design.

The automotive industry also wants smooth air flow to improve gas mileage. Complete curvature continuity also improves styling. The appearance of a car is one of the major sales factors. The potential buyer would not be impressed if the showroom lights' reflection on the car rippled and wavered. It is complete curvature continuity that makes a smooth reflection. Localized control of complex curves allows you to make minor modifications without adversely affecting the shape. For example, if a new, bigger engine wouldn't fit under a perfectly-designed hood, you could use a **NURB** spline to raise the center of the hood without changing the basic design.

NURB splines are also valuable for injection mold designs to eliminate the swirl of plastic as it is injected into the mold. Such designs provide better surface finishes and allow thinner cross-sections in the die.

Through-Points Spline Tool

The *Through-Points Spline* tool draws a spline through the points that you place.



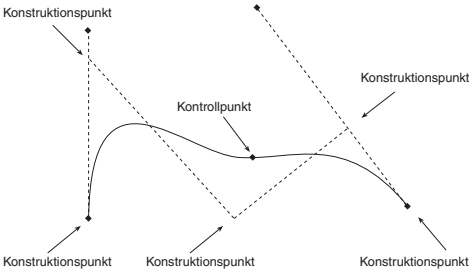
Using the Through-Points Spline tool

1. Click the points for the vectors of the spline.
2. Double-click the last point.

The status line shows the X, Y coordinates of each point as you go along.

Vector Spline Tool

This tool draws a spline using vectors determined by the points you specify. The **Vector Spline** tool uses each point that you place as the vertex of a vector for the spline it creates.



Using the Vector Spline tool

1. Click the points for the vectors of the spline.
2. Double-click the last point.

DraftBoard uses these vectors to calculate the control points (two fewer than the number of vertices you specify). The spline is tangent to the first and last vectors and passes through the calculated control points.

The status line shows the X, Y coordinates of each point as you go along.

Add Spline Control Point Tool

This tool allows you to add another control point to an existing spline.

Using the Add Spline Control Point tool

Click on the spline at the desired location(s) for the new control point(s).

Note: *To see the new control point(s), as well as the existing control points, you must first select the spline and then choose* ***Show Points*** *from the* ***Layout*** *menu (or use the* ***Edit Objects*** *dialog box to specify* ***display points****).*

If you intend to move a point, be sure to lock the adjacent points so you don't change the slope of the rest of the spline.

Lock Spline Control Point Tool

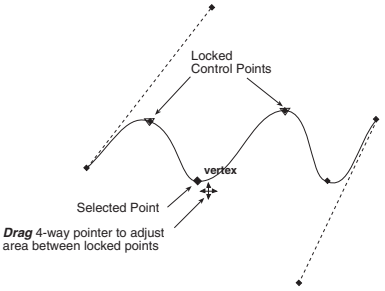
The function **Lock Spline Control Points** locks both the *control points* and the *end points* of a spline. In contrast to *control points* *spline endpoints* are automatically unlocked after each modification of the spline while *control points* remain locked until they are unlocked or another tool is selected.

This tool allows you to lock the specified points on an existing spline so you can change the slope between the points without affecting the rest of the spline.

Using the Lock Spline Control Point tool

1. Choose **Show Points** from the **Layout** menu.
2. Click the ***vertices*** you want to lock.

The **Lock Spline Control Point** tool is used to lock or immobilize selected control points on an existing spline. You may then adjust the curvature of the spline between the locked points without disturbing other areas of the spline.



Unlocking spline control points

The **Lock** and **Unlock** command in the **Arrange** menu is used to prevent changes to objects and doesn't affect spline control points.

Use this tool and click the pointer to unlock it.

Locking more than one spline control point

You can lock more than one point by simply clicking each point.

Editing control points of a spline

If you create a spline that crosses over itself, the *Drawing Assistant* will not be able to find that ***intersection***. This is by design.

1. Select the spline with the **Selection** tool.
2. Choose **Show Points** from the **Layout** menu. (If the menu says **Hide Points**, you don't need to choose it.)
3. Click on the drawing area to deselect the spline.
4. If necessary, select **Lock Spline Control Points** from the **Spline** tool subpalette.
5. If necessary, click the points you want to lock (the points on either side of the point you will change).
6. Use the **Selection** tool to select the point you want to change.
7. ***Drag*** the point to the new location.

You cannot move both ***endpoints*** of a spline at the same time. You must select and move the ***endpoints*** individually.

The dotted lines that appear tangent to the end portions of the spline are *tangent control arms*. They allow you to adjust the curvature of the spline leading up to the endpoints without moving the endpoints themselves. To do this, select the ***endpoint*** of the tangent control arm and ***drag*** it to another location with the **4-way pointer** that appears.

Editing the end slope of a spline

1. Select the spline with the **Selection** tool.
2. Choose **Show Points** from the **Layout** menu. (If the menu says **Hide Points**, you don't need to choose it.)
3. Click in the drawing area to deselect the spline.
4. If necessary, select **Lock Spline Control Points** from the **Spline** tool subpalette.
5. If necessary, select the point you want to lock but not the endpoint.
6. Use the **Selection** tool to select the point at the end of the **tangent control arm** you want to move.
7. ***Drag*** the point to a new location.

Function Plotters

The two integrated *Function Plotters* (*parametrical* and *explicit*) allow to create complex curves by calculating curve points that can be connected by a spline or line segments.

Supported mathematical Operators

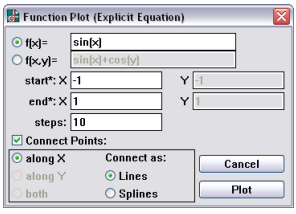
The following mathematical operators are supported by the two function plotters:

Operators	Example
Addition	3+0,5 = 3,5
Subtraction	3-0,5 = 2,5
Multiplication	3*0,5 = 1,5
Division	3/0,5 = 6
Square Root	sqrt(2) = 1,4142...
Parenthetical	3/(4+(1/0,5)) = 0,5
Exponentiation	(2^7) = 128
Sine of Angle	sin(30) = 0,5
Cosine of Angle	cos(60) = 0,5
Log (Base 10)	log(100) = 2
Absolute Value	abs(-2) = 2
Largest Smaller Integer	floor(1,4142) = 1

Function f(x);f(x,y)

The ***explicit*** *Function Plotter* calculates functions of **y** dependent on **x** or functions of **z** dependent on **x** and **y**.

When you choose the **Function f(x);f(x,y)** command in the **Modules** menu the following dialog box is displayed:



The value π must be entered into the boxes **f(x)** and **F(x,y)** as **PI** (in capital letters) and into the **Start*X** and **End*X** boxes as **PI()** (in capital letters followed by empty brackets). When defining functions only *Periods* are allowed as decimal indicators no *Commas*.

The **Function f(x);f(x,y)** dialog box allows the following settings:

f(x)= In this entry field you enter the mathematical definition for a function of **(x)**.

f(x,y)= In this entry field you enter the mathematical definition for a function of **(x,y)**.

Start*: x In the **Start*: x** entry field you enter the *Startpoint* for the ***x*** *coordinate* of the function, that you can specify with a mouse click on the drawing area.

After clicking the **Plot** button no more values can be specified with the mouse on the drawing area.

y In case the **F(x,y)=** option is marked you must enter the *Startpoint* for the ***x*** and **y** *coordinate* of the function, the you can as well specify with a mouse click on the drawing area.

End*: x In the **End*: x** entry field you enter the *Endpoint* for the ***x*** *coordinate* of the function, that you can specify with a mouse click on the drawing area.

After clicking the **Plot** button no more values can be specified with the mouse on the drawing area.

y In case the **F(x,y)=** option is marked you must enter the *Endpoint* for the ***x*** and **y** *coordinate* of the function, the you can as well specify with a mouse click on the drawing area.

Steps: In this entry field you specify the number of curve points you want to calculate.

Connect Points When this option is marked you can choose,

in case the function was defined dependent on **x**, if the calculated curve points should be connected **along x** with a **Spline** or **Line segments**,

or

in case the function was defined dependent on **x** and **y**, if the calculated curve points should be connected only **along x** or only **along y**

	or in **both** directions with a **Spline** or **Line segments**.
Plot	When you click the **Plot** button, the curve will be calculated for the specified function values and automatically plotted on the drawing area as a *Spline* or *Line segments* when the option **Connect points** is activated.

Plotting a Parabola

In the following example we will calculate and plot a parabola, that will be offset by **1** unit from the ***x*** *axis* and by **2** units from the ***y*** *axis* and will be stretch by the factor **2**.

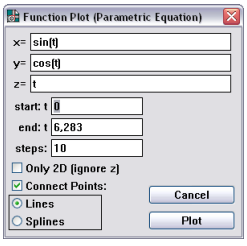
1. Choose the **Function f(x);f(x,y)** command from the Modules menu.
   The **Function f(x);f(x,y)** dialog box is displayed.
2. Select the option **f(x)=**.
3. Enter the formula **2*(x-1)^2+2** into the **f(x)=** entry box.
4. Enter the value **0** into the **Start*: x** entry field or click with the mouse a point on the drawing area for the *Startpoint* of the curve you want to calculate.
5. Enter **3** into the **End*: x** entry field or click with the mouse a point on the drawing area for the *Endpoint* of the curve you want to calculate.
6. Enter the value **12** into the **Steps** entry field for the number of curve points you want to calculate.
7. Select the option **Connect points** and the option **Splines**.
8. Click the **Plot** button.
   The specified number of curve points is calculated and the parabola is automatically plotted onto the drawing area.

Function [param]

The ***parametrical*** *Function Plotter* calculates the **x**, **y** and **z** graphs dependent on the parameter **t**.

When you choose the **Function [param]** command in the **Modules** menu the following dialog box is displayed:

The value **π** must be specified for the variables **x**, **y**, and **z** as **PI** (in capital letters) and entered into the **Start*X** and **End*X** boxes as **PI()** (in capital letters followed by empty brackets). When defining functions only *Periods* are allowed as decimal indicators no *Commas*.



The **Function [param]** dialog box contains the following elements:

x	In this entry field you enter the mathematical definition for the ***x*** *coordinate*.
y	In this entry field you enter the mathematical definition for the ***y*** *coordinate*.
z	In this entry field you enter the mathematical definition for the ***z*** *coordinate*..
Start: t	In this entry field you enter the *Startpoint* for the parameter **t**.
End: t	In this entry field you enter the *Endpoint* for the parameter **t**.
Steps: t	In this entry field you specify the number of curve points you want to calculate.
only 2D	When this option is activated the ***z*** *axis* will be ignored and a two-dimensional curve will be calculated for the ***x***, ***y*** *plane*.
Connect Points	When this option is activated you can choose if the calculated curve points should be connected with a **Spline** or **Line segments**.

Plot When you click the **Plot** button, the curve will be calculated for the specified parameters and will be automatically plotted on the drawing area as a *Spline* or as *Line segments*.

Plotting a Helix

In the following example we will calculate and plot a helix for a screw thread. The radius should be **20 mm**, the pitch of the screw thread **2 mm** and the number of threads **4**.

1. Choose the **Function [param]** command from the Modules menu.

   The **Function [param]** dialog box is displayed.
2. Enter the formula **sin(t) * 20** into the **x** entry box in which **20** equals the radius of **20 mm**.
3. Enter the formula **sin(t) * 20** into the **y** entry box in which **20** equals the radius of **20 mm**.
4. Enter the formula **2 * t /(2 * PI)** into the **z** entry field in which the first value **2** equals the pitch of the screw thread of **2 mm**.
5. Enter the value **0** into the **Start*: x** entry field.
6. Enter the formula **4 * 2 * PI()** into the **End** entry field in which the value **4** equals the number threads.
7. Enter the formula **4 * 8** into the **Steps** entry field in which the value **4** equals the number threads.
8. Select the option **Connect points** and the option **Splines**.
9. Click the **Plot** button.

   The specified number of curve points is calculated and the helix is automatically plotted onto the drawing area.

Since the initial pitch of the thread is calculated depending on the following curve points it is recommended to specify and calculate more curve points than required and to trim finally the first and the last part of the helix.

CHAPTER

3

The Drawing Assistant

Snapping onto Geometry

Drawing Assistant Construction Lines

Permanent Construction Lines

The Drawing Assistant

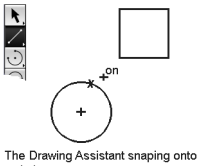
The *Drawing Assistant* is the feature that makes ***DraftBoard Unlimited*** unique among design and drafting software products. The *Drawing Assistant thinks* like a drafter; it automatically *knows* where you typically want construction lines and displays them temporarily when you need them.

The *Drawing Assistant* also makes it easy to select existing points for construction by displaying information about the pointer's location in the drawing area. If a *Drawing Assistant notation* is displayed when you click, the construction snaps onto the geometry precisely, without requiring finely tuned, eye-hand coordination or tedious selection of special modifiers, modes, or other specialized construction tools.

Snapping onto Geometry

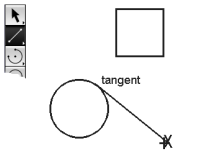
When the pointer is in the drawing area, it has a *snap point* function. The snap point locks onto specific points on existing objects as you move the pointer near them.

The *Drawing Assistant* tells you when the snap point is ***on*** an object.



The Drawing Assistant snaping onto a circle

The *Drawing Assistant* displays information about the location of the snap point. This information appears either beside the pointer or next to the object itself.



The Drawing Assistent displaying the relationship between the circle and the line that is being constructed

The *Drawing Assistant* tells you when the snap point has locked onto the points of an object.

center

The center of an arc or circle. Move the pointer across the arc or circle to display ***on*** for the arc or circle, then move the pointer near the center to display the ***center*** point notation.

endpoint

The ***endpoint*** of lines, arcs, circles, ellipses, and splines.

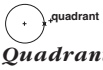
midpoint

The ***midpoint*** of lines, arcs, circles, ellipses, and splines.

intersection

The ***intersection*** of two lines or curves (both permanent lines and the *Drawing Assistant's* dynamic construction lines).

quadrant



Quadrant points on an arc or circle displayed at 3 o'clock, 6 o'clock, 9 o'clock, and 12 o'clock.

vertex

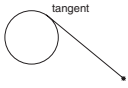
The ***vertices*** of an ellipse, spline, or dimension point.

Using tangents and perpendiculars

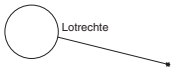
The *Drawing Assistant* locks onto a ***tangent*** or ***perpendicular*** only when the *Drawing Assistant* starts from the ***on*** notation. You cannot begin from a specific point, such as the ***endpoint***, ***quadrant*** or ***vertex*** point.

If you press a point on an arc or circle and ***drag*** the pointer away at about a 45° angle, the *Drawing Assistant* locks onto the ***tangent***. If you ***drag*** away at a 90° angle the Drawing Assistant locks onto a ***perpendicular***.

Tangent

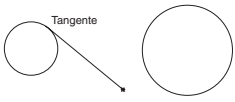


Perpendicular

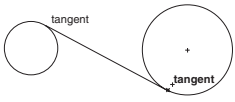


If you continue holding down the mouse button, the line remains ***tangent*** or ***perpendicular*** while you ***drag*** the ending point around the object.

This is a very useful feature if, for example, you want to create a line from and tangent to an existing circle to the tangent point of another circle.



Once a line is ***tangent*** to the circle, it can be ***dragged*** to the tangent point on the other circle, with the tangency maintained at both ends.



Keyboard Snap Points

You can direct the *Drawing Assistant* to snap onto an object. For example, you may want to start a new line from the exact center of a circle. If you hold down the mouse button and press the **c** key on your keyboard, the *Drawing Assistant* finds the ***center*** of the circle when you move the pointer near the center.

Important: *Keyboard Snap Points are not active until you have set the first construction point of any drawing or editing tool.*

Using the Drawing Assistant for snapping onto geometry

Another way would be to reduce the detection radius of the *Drawing Assistant* with a lower **number of pixels** for the hit radius or to zoom in on the drawing to separate the construction points visually (see also *Hit Radius* in this chapter).

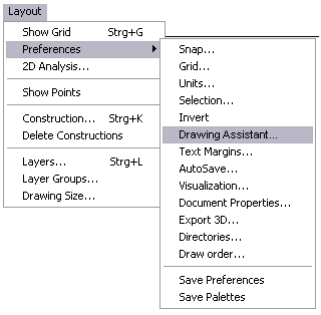
The following table lists the keys for finding specific points. The desired point must be within the hit radius of the pointer.

Letter	Snaps To (Align)
c	**c**enter
e	**e**ndpoint
g	**g**rid
i	**i**ntersection
m	**m**idpoint
n	**n**o point
o	**o**n
p	**p**erpendicular
q	**q**uadrant
t	**t**angent
v	**v**ertex
%	the percentage point set within the **Snap** dialog

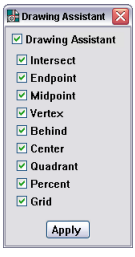
Pressing the **Spacebar** or clicking the mouse releases all snap restrictions.

Drawing Assistant

This command under **Preferences** in the **Layout** menu allows you to turn the Drawing Assistant *on* or *off* and to activate the different *snap modes* of the Drawing Assistant.



Choosing the **Drawing Assistant** command displays the following dialog box:



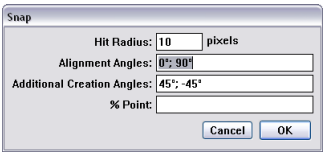
All snap modes are activated by default. If you want to deactivate a snap mode you click the related check box.

If you deactivate the *Drawing Assistant* by clicking the **Drawing Assistant** check box all snap modes are deactivated as well.

You can leave the dialog box open while drawing. You close the dialog by clicking on the **Close** box in the title bar.

Snap

This command in the **Preferences** submenu in the **Layout** menu sets specifications for the Drawing Assistant.



Hit Radius This setting determines the detection distance in pixels. When the pointer is within the specified hit radius, the *Drawing Assistant* notations are displayed and the object is selected when you click the mouse. The default hit radius is **12** pixels.

If you can't specify locations that are close together because the *Drawing Assistant* snaps to an existing *control point*, you can do any of the following:

- Decrease the hit radius.
- Zoom in so more pixels separate the existing point and the point you want to select.
- Lock on a point by pressing the mouse button and then typing the letter that represents the point (**m** for **m**idpoint, for example).

Important: *Keyboard Snap Points are not active until you have set the first construction point of any drawing or editing tool.*

Alignment Angles If you set the **Hit Radius** to zero, you disable the single click selection of the **Selection** tool (***dragging*** a marquee and double-clicking to select all objects will still work). Instead, you might consider using the **Selection Mask** in the **Edit** menu to specify that some objects cannot be selected.

These angles define the *dynamic construction* lines that the *Drawing Assistant* automatically uses. If you want to change the orientation of your drawing, you can change these specifications. For example, you could set these angles to **30°**, **90°** and **150°** for an isometric drawing. The defaults are 0° (horizontal) and **90°** (vertical). Use a semicolon to separate the values.

It is not recommended to modify the *Alignment Angles* for isometric drawings in ***DraftBoard***. Instead you should select an isometric view orientation with the *Trackball*. You find more information about isometric drawing in the *3D Part* of this manual.

To display a construction line through a point, move the pointer to the point to activate it (a diamond appears); construction lines automatically display through the active point. You can have as many as *eight* active points. When you activate the *ninth* point, the first one in the series is deactivated.

Additional Creation Angles These lines are used by the *Drawing Assistant* only when you are creating geometry and they are not part of the list of lines generated from the eight active points. The defaults are **45°** and **-45°**. Use a semicolon to separate the values.

% Point The divisions of a line for *Drawing Assistant* notations. If you want to divide the line into quarters, use a **25** specification. The default is **50**, which shows the ***midpoint*** of lines.

For example, entering *25* instructs the *Drawing Assistant* to tell you when the pointer is *25%* of the distance along a line as shown below.

Drawing Assistant Construction Lines

In addition to snapping onto geometry, the *Drawing Assistant* also displays *dynamic construction lines*. The three types of construction lines you use most frequently— vertical, horizontal, and **45°** angle lines— display automatically during construction. These construction lines appear temporarily to help you align geometry. Once you've set a point, the *Drawing Assistant* construction line disappears — your drawing is not cluttered with extraneous lines.

Dynamic construction lines extend automatically from the last point you created. You may want to activate other points so the *Drawing Assistant* displays construction lines relative to them. Simply move the pointer over the geometry to activate its control points, then move away horizontally or vertically.

Of course, you also have the option of creating *permanent construction lines* and other shapes, as described later in this chapter.

Using the Drawing Assistant's Construction Lines

If you want different construction line angles, you can change them in the **Snap** dialog box from **Preferences** in the **Layout** menu, as described in the previous section.

After you have indicated the first endpoint of a line, and you move the pointer horizontally, vertically, or in a 45° direction, the *dynamic construction* lines will appear. The figure below illustrates a 45° construction line relative to the endpoint of an existing line.

Displaying dynamic construction lines while constructing geometry

1. Click a point to begin new geometry.
2. Move the pointer horizontally or vertically.

   A construction line appears through the point.
3. While the construction line is visible and the Drawing Assistant displays ***on***, click the next point.

   The point is placed exactly on the construction line, even though the pointer wasn't exactly on that line when you clicked. The *dynamic construction* line disappears.

Displaying dynamic construction lines with existing geometry

You can have as many as **eight** active points; activating the ninth point in a series deactivates the first point.

1. Without pressing the mouse button, move the pointer over an existing point.

   The point notation (***endpoint*** or ***midpoint***, for example) shows that the point is active.

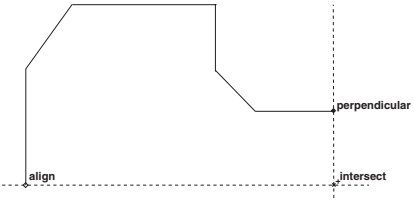
2. Move the pointer away from the point horizontally, vertically, or at a **45°** angle.

   A construction line through the point is visible.

3. While the construction line is visible and the *Drawing Assistant* displays ***on***, click the desired point in the construction.

   The point is placed exactly on the construction line, even though the pointer wasn't exactly ***on*** that line when you clicked. The *dynamic construction* line disappears.

The figure below illustrates intersecting construction lines drawn through two existing, active points.



Once you are familiar with the *Drawing Assistant*, you'll see how much ***DraftBoard*** streamlines design and drafting tasks!

Setting new Drawing Assistant construction angles

You can add to or change the angles that the *Drawing Assistant* uses for dynamic construction lines by choosing **Snap** from the **Preferences** command in the **Layout** menu. Enter the construction line angles, separated by semicolons, into the appropriate box.

Permanent Construction Lines

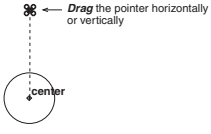
In addition to the *Drawing Assistant's* dynamic construction lines, you may want to create construction lines that remain displayed until you hide or remove them. There are two methods for creating *permanent construction lines*: strokes and the **Construction** command (in the **Layout** menu). When you want to get rid of all construction lines, choose **Delete Constructions** from the **Layout** menu. (Everything on the **Construction** layer will be deleted.)

Permanent construction lines automatically appear on the **Construction layer**, not the **work layer** of your drawing. You can hide the Construction layer to view or print the drawing without construction lines.

Stroke Construction Lines

Stroke construction lines are lines that you create with the mouse. Hold down the **Shift** and **Ctrl** key *(Macintosh: **Command** key)* and ***drag*** the mouse horizontally or vertically. Since you are working in the drawing area, the *Drawing Assistant* helps you place the stroke precisely.

Holding down the **Shift** and **Ctrl** key *(Macintosh: **Command** key)* changes the mouse pointer to the *Stroke* pointer (⌘).



You can use strokes to create construction lines while you are in the process of using a tool.

Drag	***Result***
Vertically	A vertical construction line through the first point of the stroke.
Horizontally	A horizontal construction line through the first point of the stroke.

Using Stroke construction lines

Construction lines are as long as the dimensions of the viewing area of the screen or the plot region (as designated in **Drawing Size**), whichever is larger.

For example, if you are using the **Connected Line** tool, you can create a construction line that extends through the center of a circle:

1. Hold down the **Shift** and **Ctrl** key *(Macintosh: **Command** key).*
2. Move the pointer near the center of the circle.

   The *Drawing Assistant* snaps onto the center point.

Stroke construction lines can be created only *horizontal* or *vertical*. For *construction lines* in a different angle you must use the **Construction** command in the **Layout** menu.

3. ***Drag*** the mouse vertically or horizontally away from the ***midpoint***.

   The construction line appears through the ***center*** and you are still in the process of creating connected lines.

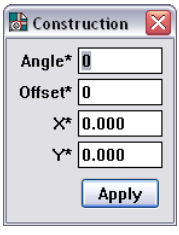
The Construction Command

Stroke construction lines are useful for creating lines through existing points. If you want to create a construction line at a location other than an existing point or at a particular angle, you can use the **Construction** command.

Construction Ctrl+K [Macintosh: ⌘ K]

DraftBoard automatically places construction lines on the **Construction** layer. When you choose **Delete Constructions** from the **Layout** menu, **everything** on the **Construction** layer is deleted, regardless of the object's pen style.

This command in the **Layout** menu creates a construction line on the **Construction** layer of the current document.



You can specify the angle of the construction line or the offset from a reference point defined by the X, Y coordinates. The asterisk shows that the values can be specified by clicking or dragging the mouse; values can also be typed into the boxes.

Specifying the construction line angle with the mouse

1. Click the **Angle** box.
2. ***Drag*** a vector in the drawing area.

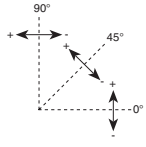
   The angle of the vector line appears in the **Angle** box.

Specifying the construction line offset with the mouse

1. Click the **Offset** box.
2. ***Drag*** the offset distance in the drawing area.

*Note: The distance **dragged** will always be entered in the **Offset** box as a positive value, regardless of the direction **dragged**.*

The distance you ***drag*** appears in the **Offset** box. Be aware that the offset is determined by the angle of the construction line as shown.



Specifying X, Y coordinates with the mouse

The coordinates of the last point you specified appear in the **X** and **Y** boxes, but you can change them by following these instructions.

1. Click the **X** box.
2. Enter new coordinates.

   Or:

   In the drawing area, click the location of the point through which the construction line should pass. The coordinates are entered automatically for both the **X** and **Y** boxes.

Before you can define a point on the drawing area you have to click in the **Entry field** where the values should go.

Creating multiple construction lines

You can create multiple construction lines through the same point by entering the angles you want separated by semicolons.

Creating parallel construction lines

You can create parallel construction lines by specifying a single angle value with different offsets separated by semicolons. You can also create parallel lines by creating one construction line using this box, then creating new lines with the **Parallel Line** tool.

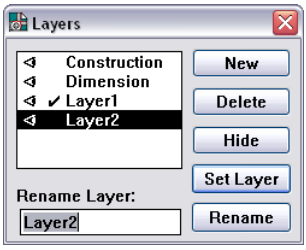
Creating Construction Geometry

Non-construction geometry is placed on the work layer. You can create temporary construction geometry, such as arcs or circles, by making

the **Construction** layer the work layer, creating the geometry and switching to another layer to continue your work. You can use the **Construction** pen style if you like, but it's not essential.

Creating construction geometry

1. Choose **Layers** from the **Layout** menu.



Lines made with the **Construction** pen do **not** go on the **Construction** layer unless you make that layer the work layer.

2. Click **Construction** in the list box.
3. Click the **Set Work** button.
4. Create the geometry you'll use for construction. You can use the **Construction** pen style, but any pen style is acceptable.
5. When the construction geometry is complete, make another layer the work layer.
6. Close the dialog box.
7. Continue your work.

Since the **Delete Constructions** command deletes all geometry on the **Construction** layer, you shouldn't create anything you want to keep on the **Construction** layer.

Once you no longer need the construction geometry, choose **Delete Constructions** from the **Layout** menu to remove all geometry on the **Construction** layer.

Removing Construction Lines

If you have many construction lines and you want to delete only one or two of them, select the lines you want to remove, and then choose the **Delete** command from the **Edit** menu or press the **Back Space** key. You can remove all the construction lines that you've created by choosing **Delete Constructions** from the **Layout** menu.

You can select construction lines in the usual manner from any layer; the **Construction** layer doesn't have to be the work layer.

Delete Constructions

This command in the **Layout** menu deletes all construction lines and any geometry on the **Construction** layer.

Since the **Delete Constructions** command deletes all geometry on the **Construction** layer, you shouldn't create anything you want to keep on the **Construction** layer.

The Drawing Assistant's *dynamic construction* lines appear only temporarily and are not affected by this command. Any geometry on the **Construction** layer (regardless of the pen style used) is deleted by this command.

You can retrieve deleted construction geometry within the limits of the **Undo** command.

CHAPTER

4

Setting the Environment

Pens

Pen Characteristics

Pen Styles

Setting the Units

Displaying the Grid

Preferences

Customization

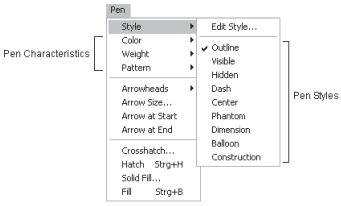
Setting the Environment

This chapter describes how to set pen styles, units and discusses the grid. The last section of this chapter tells you how to save your preferences for default settings and how you can customize ***DraftBoard***.

Pens

The pen style determines the appearance of lines on the screen and when they are plotted. If you are using a monochrome monitor or a printer, all lines will be black but the weight and pattern will be visible. Any line thickness of less than **0,07 mm** appears one pixel wide on the screen. When you print or plot such lines, you can see the different weights.

You can choose from nine different pen styles in the **Pen** menu. This menu contains as well all commands to modify all pen settings.



The current pen style is checked on the **Pen** menu. The default pen style is *Outline—solid, black lines,* ***0,35 mm*** *wide.*

You can change to a different pen style in the **Style** submenu.

You can change an individual characteristic of the current pen in the **Pen** submenus (**Color**, **Weight** and **Pattern**).

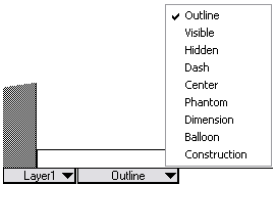
Changing the current Pen

1. Select the pen style you want to use from the **Style** submenu in the **Pen** menu.

All selected objects will be drawn in the new pen style, as will all future lines until you end the current ***DraftBoard*** session or select a new pen style.

Pen Indicator

You can also use the *Pen Indication* ***pop-up*** *box* at the lower left of the drawing area to specify the current pen.

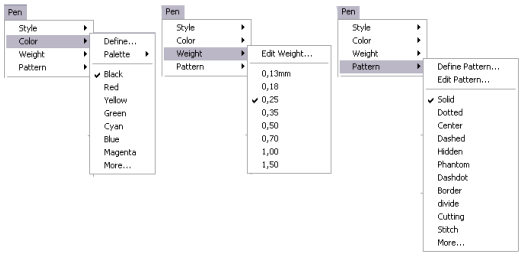


Changing the current pen with the Pen Indicator

1. Press the mouse button on the Pen Indicator box.
   The **Pens** menu displays listing all specified *Pen Styles.*
2. ***Drag*** to the *Pen Style* you want to make the current pen and release the mouse button.
   The selected pen is activated and displayed in the *Pen Indicator.*

Pen Characteristics

The following characteristics are available from the submenus in the **Pen** menu.



4 If you want to change the characteristic of a pen permanently use the **Edit Style** command from the **Style** submenu in the **Pen** menu. This command is described later in this chapter.

Whenever you change the characteristic (*Color*, *Weight* or *Pattern*) of a pen these characteristic are valid until you end the current ***DraftBoard*** session or select a new pen characteristic.

Changing one characteristic of a pen style

1. Choose the pen you want to modify.
2. Choose a characteristic—*color*, *weight*, or *pattern* – from the characteristic submenus in the **Pen** menu.

To return a standard pen to its original specifications, choose a different pen style from the menu and then choose the modified pen again.

All selected objects will reflect the new pen characteristic, as will all future lines drawn until you end the current ***DraftBoard*** session or select new pen characteristics .

Changing the pen characteristics of an object without changing the pen

1. Select the object.
2. Choose **Edit Objects** from the **Edit** menu.
   The **Edit Objects** dialog is displayed.
3. Change the pen characteristic in the **Edit Objects** dialog box.
4. Click the **Apply** button.
   The selected objects reflects the new pen characteristics. The characteristics of the current pen are not modified.

Colors

The **Color** submenu in the **Pen** menu contains **234** color options from a palette of *16.7 million* colors available. **64** colors are predefined, **170** colors are user definable.

You select a new color for the current pen from the **Color** submenu in the **Pen** menu where you find also the command to define your own colors.

Color

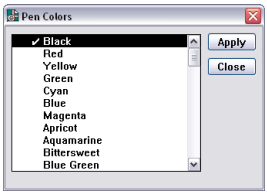
This command in the **Pen** menu specifies the **color** of the current pen and any selected geometry without changing any other pen characteristics.

You can change the color of the current pen in the **Color** submenu by selecting a *color name* or by selecting a new color from the *Color subpalette*.

Specifying the color of the current pen by selecting a color name

1. Select the **Color** command from the **Pen** menu.
   The submenu appears.
2. ***Drag*** to the color you want.
   The current pen and all selected objects takes on the new color and the color is selected in the submenu.

3. Select **more** from the **Color** submenu, if you want to select a different color than one of the seven displayed basic colors.
   The **Color** dialog box is displayed listing all available colors.



4. Choose a color name.
5. Click the **Apply** button
   The current pen and all selected objects takes on the new color.
6. Click the **Close** button when you have modified all line colors.

Choosing the color of the current pen from the color palette

1. Select the **Color** command from the **Pen** menu.
   The submenu appears.
2. ***Drag*** to **Palette** in the **Color** submenu.
   The **Color Palette** is displayed showing 64 predefined and all user defined colors.

3. ***Drag*** to the color you want.
   The current pen and all selected objects takes on the new color and the color is selected in the submenu.

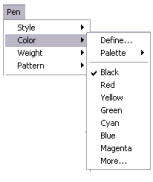
Changing the pen color of an object without changing the color of the current pen

1. Select the object.
2. Choose **Edit Objects** from the **Edit** menu.
   The **Edit Objects** dialog is displayed.
3. Change the pen color in the **Edit Objects** dialog box.
4. Click the **Apply** button.
   The selected objects reflects the new pen color. The color of the current pen is not modified.

Define Colors

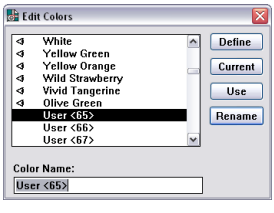
You can define **170** colors in addition to the **64** predefined colors.

New colors are defined with the **Define** command from the **Color** submenu in the **Pen** menu.



Specifying a new color name

1. Choose the **Define** command from the **Color** submenu.
   The **Define Color** dialog box is displayed.



The **Define Color** dialog box displays a list containing all color names that can be defined in ***DraftBoard***. The list starts with the *64 predefined* colors followed by the *170 user definable* colors named **Color <65>** to **Color <236>**.

2. Scroll down the list and select the first undefined color number, like **Color <65>**.
3. Enter the color name (limited to 15 characters) in the **Color Name** data field.
4. Click the **Rename** button.

   The undefined **Color <65>** is renamed.

Now we specify a color for the new color name.

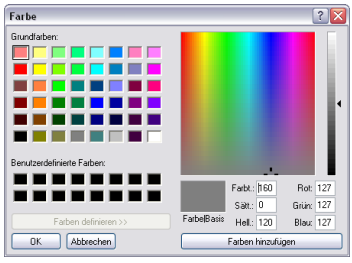
WIN

Defining a color

WIN

1. Click the **Define** button in the **Define Color** dialog box.

   The *Windows* ***Color*** *dialog box* is displayed.



2. Click the **Define Custom Color** button.

   The extended **Color Palette** is displayed.
3. In the color palette on the right, place the cursor within the target cursor and ***drag*** to the area of the color desired. Note a larger area of the color is displayed in the **Color/Solid** box below.

   If desired, you can also adjust the *hue*, *saturation*, *luminance*, and the percentage of *red*, *green*, and *blue* by entering your own values.

   The % of black can be adjusted by moving the cursor over the arrow in the color scroll bar to the right of the large color palette and drag up or down to the desired location. When you adjust this, all the values in *Luminance*, *Red*, *Green*, and *Blue* data fields change as well. *Hue* and *Saturation* do not change.
4. Click the **Add to Custom Colors** button.

   The defined color is displayed in the first highlighted box for custom defined colors.
5. Click the **OK** button.

   Die *Windows* ***Color*** dialog box closes and you are returned to the **Define Colors** dialog box.
6. Click the **Use** button in the **Define Colors** dialog box.

   An *icon of an eye* will appear, indicating that the color will be visible now in the ***DraftBoard*** *Color Palette*. The new color name remains in the color list in the position you placed it, since the color names are not alphabetized.
7. Close the **Define Color** dialog box.

MAC

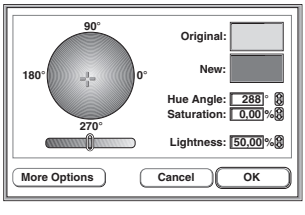
Defining a color

MAC

1. Click the **Define** button in the **Define Color** dialog box.

   The *Apple* ***HSL*** *Color Wheel* is displayed.

The *Apple Macintosh* offers two system to define colors: The *Apple* ***HSL*** *Color wheel* and the *Apple* ***RGB****-Color display*. You can switch between the two systems or use one system as your default color system.



The *Apple* ***HSL*** *Color Wheel* shows the complete color range of all colors available.

2. ***Drag*** the mouse pointer in the color wheel to the area of the color desired.

   The selected color is displayed in the **New** box below the **Original** box and its color values are displayed in the boxes for *hue angle*, *saturation*, and *lightness*.

When you define a color for the first time the color ***grey*** is displayed in the Original color field.

3. Modify the values for *hue angle*, *saturation*, and *lightness*. The lightness can be adjusted as well with the ***Lightness*** *Slide bar* below the *Color Wheel*.

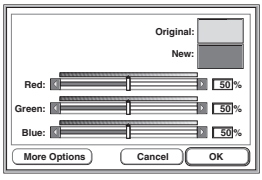
4. Click the **More Options** button.

   On the left hand side the icons for the *Apple* ***HSL****-Color wheel* and the *Apple* ***RGB*** *Color display* are displayed.

The icons for the two Apple color systems can be hidden using the **Fewer Options** button.

5. Click onto the icon for the *Apple* ***RGB*** *Color display* when you want to specify the new color by its basic **RGB** Colors *Red*, *Green* and *Blue*.

   The *Apple* ***RGB*** *Color display* is displayed.



6. Specify the color using thew ***Color*** *Slide Bars* for *Red*, *Green* and *Blue*.

   The specified color is displayed in the **New** box below the **Original** box and its color values are displayed in the boxes for the colors *Red*, *Green* and *Blue*.

7. Click the **OK** button.

   The *Apple* ***HSL****-Color wheel* or the *Apple* ***RGB*** *Color d*isplay – depending which system was displayed– will be closed and you are you are returned to the **Define Colors** dialog box.

8. Click the **Use** button in the **Define Colors** dialog box.

   An *icon of an eye* will appear, indicating that the color will be visible now in the ***DraftBoard*** *Color Palette*. The new color name remains in the color list in the position you placed it, since the color names are not alphabetized.

9. Close the **Define Color** dialog box.

Saving user defined colors

User defined colors are automatically saved with the drawing as soon as you save the drawing using the **Save** command from the **File** menu.

To save user defined colors permanently choose the **Save Preferences** command from the **Preferences** submenu in the **Layout** menu.

Arranging the color palette

All *user defined colors* are automatically displayed in the ***DraftBoard*** *Color Palette* when an *eye symbol* is displayed in front of the color name in the **Define Color** dialog box.

If there are any undefined colors between the last *predefined color* (**Color <64>**) and the last *user defined color* all these undefined colors are shown in the color palette by a cross.

If you want to have a continuous color palette you must define all color continuously after the last predefined color.

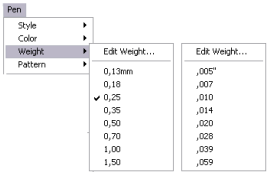
Pen Weights

The **Weight** submenu in the **Pen** menu offers **8** pen weights, that are all user definable. This submenu contains also the **Define** command, that allows you to specify for each of the pen weighs displayed in that menu a specific weight.

Weight

This command in the **Pen** menu sets the pen **width** of the current pen and selected lines without changing any other pen characteristics.

The line weights shown in this submenu are dependent on the selected **Units** in the **Preference** Submenu. If you have selected metric units, the line weight will be shown in **mm** instead of **inches**.



Specifying a new weight for the current pen

1. ***Drag*** to **Weight** in the **Pen** menu.
   The submenu appears.
2. ***Drag*** to the weight you want.
   The pen takes on the new weight and the weight is selected in the submenu.

Any line thickness of less than **0,07 mm** appears one pixel wide on the screen. When you print or plot such lines, you can see the different weights. The *Line weight* of *dimension lines* can be only modified in the **Preferences** submenu of the **Dimension** menu and should be always **0,25 mm**.

4

Edit Weight

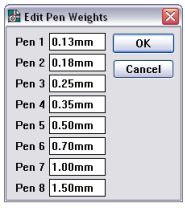
This command from the **Weight** submenu in the **Pen** menu sets the weights for ***DraftBoard's*** lines. ***DraftBoard*** comes with eight pens with differing line weights ranging from **0.13** to **1.50** mm. Objects drawn in ***DraftBoard*** are drawn in one of the eight available pen weights. With the **Edit Weight** command in the **Pen** menu, you can change the weight of one of the pens to any value between **0.00 mm** to **1.50 mm**.

There cannot be more than eight different pen weights in a drawing, so changing a pen weight in the **Edit Pen Weights** dialog box will change the weight of every object that was drawn with that pen's former weight.

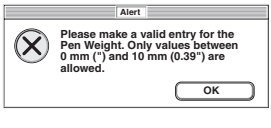
Editing Line Weights

1. Choose **Edit Weight** from the **Weight** submenu of the **Pen** menu.
   The **Edit Pen Weigh**t dialog box appears.

Remember that *pen weights* thinner than .007 mm will show up on the screen as one pixel thick. However, you should be able to see the difference in *pen weights* on your prints and plots.



2. Select the pen weight you want to change by clicking on the pen number. That pen's entry box should be selected.
3. Change the pen weight by typing a new value into the entry box. If no units are given with the entry, ***DraftBoard*** applies the unit specified in the **Units** dialog box from **Preferences**.
   If the entry does not fall with in the valid range from **0** to **0,039** Inch or **10 mm**, ***DraftBoard*** prompts you with an alert box similar to the following one.



4. Click **OK**.
   All existing lines drawn in the pen weight you edited will be changed to reflect the new weight, as will all future lines will be drawn in that weight. The new pen weights are valid until the end of the current ***DraftBoard*** Session or you define another pen weight. Clicking **Cancel** ignores all changes made to any pen weights and closes the **Edit Pen Weights** dialog box.

Undoing a Pen Weight Edit

Edited pen weights are not backwards compatible with previous versions of ***DraftBoard***. Opening a file that contains edited pen weights in a previous version of ***DraftBoard*** will revert the pen weights back to their original settings.

You cannot undo editing a pen weight with the **Undo** command. To return a pen back to ***DraftBoard's*** default, you will need to enter the original value in the **Edit Pen Weight** dialog box, following the steps described above.

Saving edited Pen Weights permanently

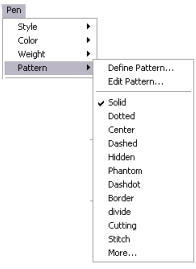
Edited pen weights affect only the pen weights in the current file. To save the edited pen weights so that they are available in all future files, choose the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

Line Patterns

DraftBoard Unlimited allows to specify **19** line patterns in addition to **11** predefined line patterns.

Pattern

This command in the **Pen** menu sets the **Pattern** of the current pen and any selected lines without changing any other pen characteristics.



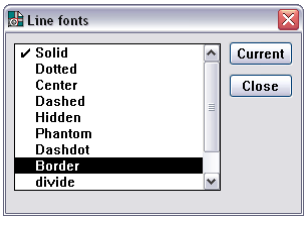
The **Pattern** submenu in the **Pen** menu contains all defined line patterns. **11** different line patterns can be selected directly, for all others you have to display a dialog box using the **More** command.

Specifying line patterns in the Pattern submenu

1. ***Drag*** to **Pattern** in the **Pen** menu.
   The submenu appears.
2. ***Drag*** to the pattern you want.
   All selected objects take the new pattern and all new objects are drawn in this line pattern.

Specifying line patterns with the More command

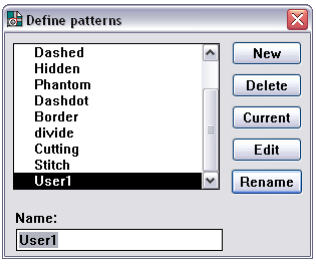
1. ***Drag*** to **Pattern** in the **Pen** menu.
   The submenu appears.
2. Select the **More** command.
   The **Line Fonts** dialog window is displayed.



3. Select the pattern you want.
4. Click the **Current** button.
   All selected objects take the new pattern and all new objects are drawn in this line pattern.

Defining Line Patterns

In ***DraftBoard*** you can specify in addition to the ***11*** *predefined* ***Line Pattern*** another **19** using the Line Editor. The *Line Editor* is displayed with the **Define Pattern** command in the **Pattern** submenu of the **Pen** menu.



The **Define Patterns** dialog box contains the following elements:

New When you click this button, a new line pattern with the name **User 1** is created, that you can rename using the **Rename** button. You can specify the new pattern us-

ing the *Line editor*, that you display with the **Edit** button.

Delete This button allows you to delete the line pattern that is selected in the **Define Pattern** dialog box. You can only delete user defined line patterns. The ***11 predefined Line patterns*** cannot be deleted.

Current This button makes the selected line pattern the current pattern. All selected objects take the new pattern and all new objects are drawn in this line pattern.

Edit This button displays the *Line Editor*, that allows you to edit the new pattern. You can edit all line patterns except the two predefined line patterns ***solid*** and ***dotted***.

Rename This button allows you to rename a line pattern with the name entered into the **Rename** field.

You can only rename user defined line patterns. The ***11 predefined Line patterns*** cannot be renamed.

Defining Line Patterns

1. Select the **Define Pattern** command in the **Pattern** submenu in the **Pen** menu.

   The **Define Patterns** dialog box is displayed.
2. Click onto the **New** button.

   A new line pattern with the name **User 1** is created.
3. Rename the new line pattern in the **Rename** field with a clear name.
4. Click the **Rename** button.

   The new line pattern is renamed with the name you entered into the **Rename** field.
5. Click the **Edit** button.

   The **Edit Pattern** dialog box (*Line Editor*) is displayed.
6. Specify a pattern for the new line pattern.
7. When you have specified the new line pattern, click the **OK** button.

   The dialog window will be closed and the specified pattern is saved as new line pattern.
8. Click the **Current** button, when you want to make the new pattern the current pattern.

   All selected objects take the new pattern and all new objects are drawn in this line pattern.

Editing Line Patterns

With the **Edit Pattern** command from the **Pattern** submenu in the **Pen** menu you can set the spacing for ***DraftBoard's*** line patterns. ***DraftBoard*** automatically scales line patterns by the inverse of the scale factor set in **Drawing Size** or **Sheet Into View** so that the pattern spacing will be appropriate for the viewing and drawing scale of the geometry. Sometimes, though, you might want to set the pattern spacing independently of the viewing or drawing scale, or change the pattern altogether.

The **Edit Pattern** dialog box lets you control the pattern spacing for line patterns.

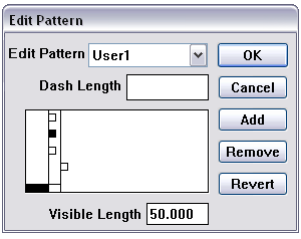
In ***DraftBoard Unlimited*** you can edit all line patterns except **2** (*solid* and *dotted*) of the **11** predefined line patterns.

That means that changing, for instance, the **Phantom** line pattern in the **Edit Pattern** dialog box will change the pattern of every line in the existing document that is drawn in the **Phantom** line pattern.

All line patterns are edited in the *Line Editor* that you display using the **Define Patterns** command in the **Pattern** submenu of the **Pen** menu.

Edit Pattern

When you choose **Edit Pattern** from the **Pattern** submenu of the **Pen** menu. The **Edit Pattern** dialog box appears.



The **Edit Pattern** dialog box contains the following settings:

Edit Pattern In this pull-down list you select the line pattern you want to change. The list box contains nine line patterns that can be modified.

Pattern Window ***DraftBoard*** considers each line segment and each space between line segments separate Dashes, and each Dash has a handle, represented by a box attached to the vertical line extending from the end of each Dash. One complete element of the line pattern appears in the *Pattern Window*.

Visible Length Displaying one complete element of the line pattern results in a specific visible length of the displayed pattern. The default visible length of *50 mm* allows to displays all predefined line patterns of ***DraftBoard***. If you want to create patterns with Dashes or elements larger than *50 mm*, change the value in the **Visible Length** entry box to an appropriate number; the *Pattern Window* will scale the pattern element accordingly.

Dash Length There are two ways to change the length of a Dash.

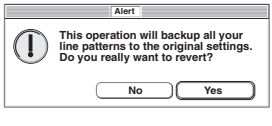
- Clicking the Dash handle activates it—the handle is now filled. The Dash's current length appears in the **Dash Length** entry box. Entering a new value will change the length of the selected Dash.
- You can also ***drag*** the handle with the mouse. As you drag, the Dash's length is updated in the **Dash Length** entry box. Release the mouse button when the Dash is the length you want.

Add Using the **Add** button you can add a new Dash. The new Dash, with a length of zero, is active; its handle is activated. You can change the length of the new Dash by ***dragging*** its handle or by typing in a value in the **Dash Length** entry box.

Remove Using the **Remove** button you can remove a Dash. To remove a Dash, select its handle and click **Remove**. The Dash is removed from the pattern element.

Revert You cannot undo a pattern edit with the **Undo** command. To return a pattern back to ***DraftBoard's*** default, click **Revert** in the **Edit Pattern** dialog box.

Revert changes *all* the line patterns back to their defaults, not just the pattern selected from the **Edit Pattern** pull-down list. A warning message is presented when you click **Revert**.



Clicking **Yes** will change *all* of the patterns back to ***DraftBoard's*** original settings, and clicking **No** will return you to the Edit Pattern dialog box without returning any patterns back to their original settings.

OK When you click **OK** all existing lines drawn in the line pattern you edited will be changed to reflect the new pattern element, as will all future lines drawn in the line pattern.

Cancel Clicking **Cancel** ignores the all changes made to any pattern elements and closes the **Edit Pattern** dialog box.

Editing a Line Pattern

1. Choose **Edit Pattern** from the **Pattern** submenu of the **Pen** menu.

   The **Edit Pattern** dialog box appears.

2. Select the line pattern you want to change from the **Edit Pattern** pull-down list.

3. Modify the line pattern by changing the lengths of existing Dashes either by ***dragging*** its handle or by typing in a value in the **Dash Length** entry box. Add a new Dashes by clicking the **Add** button or remove dashes by selecting its handle and clicking the **Remove** button.

4. When the pattern element is properly defined, click **OK**.

   All existing lines drawn in the line pattern you edited will be changed to reflect the new pattern element, as will all future lines drawn in the line pattern.

   Clicking **Cancel** ignores the all changes made to any pattern elements and closes the **Edit Pattern** dialog box.

4 Edited Patterns are affect only the patterns in the current file. To save the edited pattern so that it is available in all future files, choose the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

Undoing a Pattern Edit

1. Click the **Revert** button in the **Edit Pattern** dialog box.

   A message is displayed warning you that clicking **Revert** changes *all* the line patterns back to their defaults, not just the pattern selected from the **Edit Pattern** pull-down list.

2. Click the **Yes** button

   A*ll* of the patterns are changed back to ***DraftBoard's*** original settings.

You cannot undo a pattern edit with the **Undo** command in the **Edit** menu.

Clicking **No** will return you to the **Edit Pattern** dialog box without returning any patterns back to their original settings.

Pen Styles

The following styles are the default settings of those pens that are available in the **Pen** menu:

Outline

Pattern: **Solid**, Weight: **0.35 mm**, Color: **Black**. This is the default pen setting.

Visible

Pattern: **Solid**, Weight: **0.5 mm**, Color: **Black**.

Hidden

Pattern: **Hidden**, Weight: **0.35 mm**, Color: **Red**. Hidden lines are actually visible on the screen and in your drawing; they represent lines that would be hidden in a solid object.

Dash

Pattern: **Dashed**, Weight: **0.35 mm**, Color: **Yellow**.

Center

Pattern: **Center**, Weight: **0.35 mm**, Color: **Green**.

Phantom

Pattern: **Phantom**, Weight: **0.35 mm**, Color: **Cyan**.

Dimension

Pattern: **Solid**, Weight: **0.25 mm**, Color: **Blue**. Dimension lines are used for all type of dimension.

Balloon

Pattern: **Solid**, Weight: **0.25 mm**, Color: **Green**.

Construction

Pattern: **Dotted**, Weight: **0.25 mm**, Color: **Magenta**.

The *Construction* line style doesn't create *dynamic* or *permanent Construction Lines*. This line style uses the same line pattern but is not automatically placed on the **Construction** layer and can't be deleted with the **Delete Constructions** command from the **Layout** menu.

Modifying Pen Styles

You can change the characteristics of a pen the **Edit Style** command in the **Pen** menu.

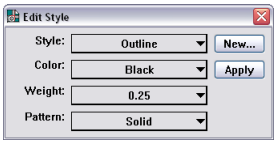
Edit Style

This command in the **Pen** menu sets the characteristics for the pen styles.

Redefining the specification of a pen style

1. Choose **Edit Style** in the **Pen** menu.

   The **Edit Style** dialog box appears.



2. Select the pen style you want to change by pressing the **Style** entry box. When the list of pen styles appears, ***drag*** to the style you want to change.
3. Specify the characteristics (*color*, *weight*, and *pattern*) you want for that style.
4. Click the **Apply** button.

   The selected pen style has now the new specifications until you quit ***DraftBoard***. When you click the **Apply** button all selected objects in the drawing area obtain the new specifications. The dialog box stays open so you can make changes to other pen styles.

When you edit a style, you set new specifications for future uses of that pen. The specifications remain in effect for the document in which they are set until you change it again with the **Edit Style** command.

You can change the default setting of any pen style by saving changes in the preferences file. See *Saving Preferences* later in this chapter.

Redefining the specification of a single pen style

When you only want to modify the characteristics of a pen as long as you use this pen you must proceed as follows:

1. Select the pen style you want to change
2. Specify the characteristics (*color*, *weight*, and *pattern*) you want for that style.

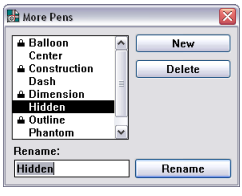
When you want to restore the original characteristics of the pen you select any other pen and then again the pen whose attributes you have modified.

Defining new Pen Styles

1. Choose the **Edit Style** command from **Style** in the **Pen** menu.

   The **Edit Style** dialog box is displayed.
2. Click the **More** button.

   The **More Pens** dialog box is displayed.



3. Click the **New** button.

   A new line with the default name **Line 1** is created, that you can rename using the **Rename** button.
4. Click in front of the name of the new line in case you want to lock the line to prevent any modification.

   A lock in front of the name is displayed. If you want to unlock the line again you must click on the lock in front of the line name.
5. If necessary create or rename more pens.

   The pens *Dimension*, *Balloon*, *Outline* and *Construction* are permanently locked and can't be modified.
6. Close the **More Pens** dialog box.
7. Select the new pen by pressing the **Style** entry box. When the list of pen styles appears, ***drag*** to the new pen you want to change.
8. Specify the characteristics (*color*, *weight*, and *pattern*) you want for the new pen.

9. Click the **Apply** button.
   The new pen has now the specified attributes for the current document until you quit ***DraftBoard***. When you click the **Apply** button all selected objects in the drawing area obtain the new specifications. The dialog box stays open so you can make changes to other pen styles.

All *pen attributes* specified in the **Edit Style** dialog box are saved with the drawing and are still valid when the document is opened again.

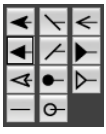
Arrow Lines

With the commands **Arrow at Start** and **Arrow at End** from the **Pen** menu you can specify the placement of *Arrowheads* on lines or circular arcs that are not a part of dimensions. You can choose the Type of arrowhead from the **Arrowheads** submenu of the **Pen** menu. The size of the arrowhead can be specified in the **Arrow size** submenu of the **Pen** menu.

Arrowheads

The **Arrowheads** submenu of the **Pen** menu allows you to choose one of eight arrowheads for arrow lines.

Arrowheads for *dimension lines* are defined separately under **Preferences** in the **Dimension** menu.

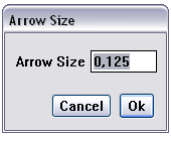


The arrowhead selection is valid until the end of the current ***DraftBoard*** session. You can save the selected *Arrowhead* permanently with the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

Arrow size

The size of an arrowhead is specified in the **Arrow size** submenu of the **Pen** menu. Choosing this command displays the following dialog box:

The size of dimension arrowheads is set separately under **Preferences** in the **Dimension** menu.

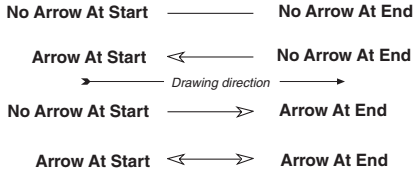


To save the size of an *Arrowhead* permanently you must choose the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

For the arrow size you can enter a factor in the **Arrow size** entry box. The default factor is **3,5**. A modified arrow size is valid until the end of the current ***DraftBoard*** session.

Arrow at Start

This command in the **Pen** menu places an arrowhead at the beginning of selected and subsequent lines and circular arcs. You can choose the type of arrowhead from the **Arrowheads** submenu of the **Pen** menu. A checkmark (■) indicates the current arrowhead setting.



Arrow at End

This command in the **Pen** menu places an arrowhead at the end of selected and subsequent lines and circular arcs. You can choose the type of arrowhead from the **Arrowheads** submenu of the **Pen** menu. A check mark indicates the current arrowhead setting.

Creating arrow lines or arcs

1. Select one or several lines you want to place an arrowhead at the beginning or at the end.
2. Choose an arrowhead from the **Arrowhead** submenu of the **Pen** menu.

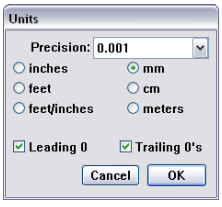
3. Select **Arrow at Start** or **Arrow at End** in the **Pen** menu. If you select both commands arrowheads are placed at both ends of the selected lines or arcs.
4. Use the **Arrow size** command of the **Pen** menu to adjust the size of the placed arrowheads.

Setting Units

DraftBoard is set to measure geometry in millimeters. When you open a new drawing you should set the precision, units, and fractional or decimal specifications, to suit your needs.

Units

This command is in the **Preferences** submenu in the **Layout** menu.



When you change the units, existing dimensions update to reflect the change, converting automatically between English and metric.

Precision Sets the number of decimal places in all data fields within ***DraftBoard*** (in *Edit Objects*, the *Status line*, and all *Dialog boxes*). The default is **0,01**.

When you choose a **fractional** precision, measurements will appear as fractions rather than decimals. You can set the format for fractions in the **Linear** submenu in the **Dimension** menu.

When ***DraftBoard*** rounds off values it stores the 16-digit decimal number; therefore, you can change the precision displayed at any time and maintain accuracy.

Altering the precision does not affect any dimensions. The precision of dimensions is selected in the **Preferences** submenu of the **Dimension** menu.

Inches, feet Displays measurements in English units. Measurements less than *6 feet* appear in inches and those greater than *6 feet* appear in feet and inches.

mm, cm, meters Displays measurements in metric units.

Leading 0 Determines whether or not the zero to the left of the decimal point appears.

Trailing 0's Determines whether or not zeros to the right of the decimal point are displayed. If set, the precision determines the number of trailing zeros.

The options **Leading 0** and **Trailing 0's** set only the number of decimal places in all data fields (*Edit Objects*, *Status Line*, *Position Indicator*) and in dialog boxes like **Edit Objects**, **2D Analysis** etc.). Leading **0** and **Trailing 0's** for dimensions are set in the **Preferences** submenu of the **Dimension** menu.

Using the status line to specify the unit of measure

You can use the status line to specify a unit of measure other than the one in effect for the document. If the units are set to inches and you draw a line, you can use different units of measure such as inches ("), feet ('), feet and inches (x'y"), millimeters (mm), centimeters (cm), and meters (m). You can also mix the units in the mathematical expression as long as you label the units properly-for example, 10" + 25.4 cm.

In the following example, if the units are set to **Meter**, and a line is drawn using the value below, a line **23 cm** long will be drawn when you press the **Enter** key, and the status line will show the measurement in meters.

In addition, you can use *mathematical*, *exponential*, and *trigonometric* expressions in the status line.

Displaying the Grid

When the grid is visible, objects snap to the grid spacing. If you want to place an object between the grid marks, you must either turn off the grid, change the grid spacing, or zoom in so that the spacing is larger than the hit radius. The grid spacing automatically reflects the units you set in the **Units** dialog box. When you change the units, the appearance of the grid spacing remains the same, and the values listed in this dialog box change automatically.

Grid

This command in the **Preferences** submenu on the **Layout** menu allows you to set the spacing of the grid lines. You can specify the number of ticks (**subdivisions**) per unit (**spacing**). You can change the default setting by saving changes in the preferences file.

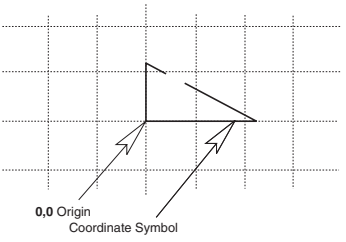
Show Grid Ctrl+G [*Macintosh*: ⌘G]

When you make any changes in the **Grid** dialog box, the grid is automatically displayed after closing the **Grid** dialog box.

This command on the **Layout** menu controls the display of the grid. The grid is made up of horizontal and vertical lines of dots. The spacing of the grid lines and the number of tick marks between the intersections are set in the **Grid** dialog box which you can choose from the **Preferences** command on the **Layout** menu.

When the grid is visible, the *Drawing Assistant* snaps to the grid spacing. In other words, if the grid is set to *25 mm* spacing, you can't construct an object closer than *25 mm* to another object. The coordinate symbol appears at the origin location (**X=0, Y=0**) when the grid is visible.

Displaying the *Grid* is rarely required, since the *Drawing Assistant* offers in most cases a better support than the *Grid*.



Saving Preferences

All files are saved with the settings as they are established when you save and close the file. The characteristics used for new files (the default settings) are contained in the *prefs.vlm (Macintosh:* ***DraftBoard Prefs****)*. The file must be stored in the same directory as the ***DraftBoard*** application.

Preferences

This command in the **Layout** menu displays a submenu for setting various specifications. Descriptions of the items in the submenu appear individually in this reference section. You can change the default setting by saving changes in the preferences file, ***prefs.vlm*** *(Macintosh:* ***DraftBoard Prefs****)*.

Snap	Use this dialog box to set the alignment angles, hit radius, and other settings for the *Drawing Assistant*.
Grid	This dialog box allows you to specify the spacing of the grid lines. Use the **Show/Hide Grid** command in the **Layout** menu to turn the display of the grid on and off.
Units	This dialog box specifies the units and precision for the current ***DraftBoard*** document.
Selection	Use this dialog box to set the color or blinking which shows that an object is selected.
Invert	This commands changes the *Background color* (the color of the drawing area from *white* to *black*)
Text Margins	In this dialog box you can specify values for the *Left*, *Right*, *Top* and *Bottom* text margins.
AutoSave	This dialog box allows the user to set a time interval for the program to save files.
Document Properties	In this dialog box the user can set document properties such as *Author*, *User defined attributes* or to automatically create a document *Preview*.
Drawing Assistant	In this dialog box you can turn the *Drawing Assistant On* or *Off* and activate different *snap modes*.
Directories	This dialog box allows the user to specify directories for the *Open*, *Save* or *Import* command.
Draw Order	This command activates the **Draw Order** that will display all objects according to the **Bring to Front/**

Bring to Back command when redrawing the screen. In complex drawings switching this command off can accelerate the screen redraw.

Visualization This dialog box gives the user the ability to choose settings for shading, background color, hidden line appearance and the use of the default lighting.

Save Preferences If you have set the characteristics you like, such as units, pen style and dimension format, you can make those settings the default setting. Choose **Save Preferences** from the **Preferences** submenu on the **Layout** menu. You cannot save geometry in this way; you would have to open the preferences file, create the geometry, and save the file.

Save Palettes When you select this command the *Status* (visible/hidden) and the *Position* of all palettes are saved to preferences. When you restart ***DraftBoard*** all palettes that were displayed when you closed ***DraftBoard*** are automatically reopened.

Position and *Status* for the following palettes are saved with this command:

- Dimension palette
- Symbol palette
- Function palette
- BOM palette

Status and *Position* of these palettes are saved in the **DraftBoard.ini** file under the section **Palettes**.

If a palette is not visible it could be that it was saved at a position outside of the screen. In this case you look for the name of the palette in the *DraftBoard.ini* file in the ***DraftBoard*** directory or folder under the section Palettes and enter there the Pixel Coordinates 20, 20. Then you have to re-start ***DraftBoard***.

The preferences file name is *prefs.vlm (Macintosh:* ***DraftBoard Prefs****)*. The file must be stored in the same directory as the ***DraftBoard*** application.

If you want to use different settings for your work, you can change the default settings, and then every new document opens with the settings you have selected. The following specifications can be set in the preferences file:

- Pen styles
- Text characteristics
- Text Margins
- Preferences settings (*Snap*, *Grid*, *Units*, and *Selection* indicator, *Auto Save*, *Document Properties* and *Directory* settings)
- Grid display
- Layer and sheet specifications
- Work layer
- Dimension and tolerance formats
- Arrowhead type and display
- Drawing size and scale
- Zoom scale
- Fillet radius
- Chamfer angle and length
- Parametric Resolve values
- User Defined Colors
- Visualization Parameter for ***3D*** *Models*

You cannot save geometry into the preferences file with the **Save Preferences** command, only your settings are saved. Be careful to use the **Save Preferences** command when your drawing contains several *Sheets*, *Models*, *Detail Views* and *Layers*. Although this command doesn't save any geometry, it saves the setup of the current drawing as default for all future drawings you create. That means that any new drawing will contain the same number of *Layers*, *Detail Views* or *Models* as the drawing you used the **Save Preferences** command with. Therefore check regularly your *Preference File*. It should contain per default ***1*** *Sheet*, ***1*** *Model*, ***no*** *Detail View* and ***4*** *Layers* (Layer 1, Construction, Kinematic and Dimension).

Changing the default settings

1. Create a file which has the preferences you want.
2. Choose **Save Preferences** from the **Preferences** submenu in the **Layout** menu.
   The preferences are set for subsequent new documents.

The **Save Preferences** command saves as well *3D Parameters* that are discussed in detail in the *3D Part* of this manual.

Customization

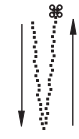
DraftBoard can be customized to your personal needs by displaying a **Floating Tool Palette**, creating **Macros** or adding commands and functions via the **Function Interface**. Both the *Floating Tool Palette* and *Macros* can be executed by ***Stroke*** commands. Customizations can also be made by changing the contents of the *DraftBoard.ini* file.

Floating Tool Palette

DraftBoard's standard Tool palette is fixed at the left side of the ***DraftBoard*** *window*. But you may display an additional **Floating Tool Palette** which you can ***drag*** and place anywhere on the screen.

Displaying the Floating Tool Palette

1. Press and hold the **Control** and **Shift** keys (*Macintosh:* ***Command key***).
2. Depress the mouse button and ***drag*** the mouse pointer first down and then up.



Drag the pointer first down then up.

3. Release the **Ctrl** and **Shift** keys (*Macintosh:* ***Command key***).
   The **Floating Tool Palette** is displayed.

You can close the **Floating Tool Palette** by clicking the **Close** box in the upper left corner of the *Floating Tool Palette*.

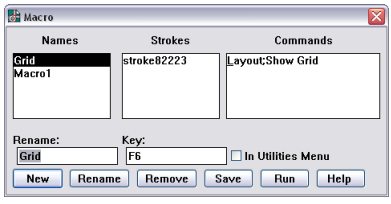
Macros

The **Macro** capability of ***DraftBoard*** allows you to create macros and access them through *menu commands*, *stroke commands*, or *key combinations* providing an alternative way to invoke ***DraftBoard*** commands.

A macro automates a repetitive task by executing a group of simple commands, If you can point and click, you can define a macro. That's how easy it is!

Macro

If you call the **Macro** command in the **Utilities** menu the following dialog box is displayed:



The **Macro** dialog box is modeless in the sense that there are no **Start Recording** or **Stop Recording** buttons. Whatever action you perform (selecting menu items or defining strokes) will be recorded automatically in the relating list boxes as long as the **Macro** dialog box is displayed.

The **Macro** dialog box contains the following buttons:

New — Clicking the **New** button adds a macro named **Macro1** to the **Names** list box. You can rename the macro in the **Rename** field by overwriting **Macro1** with a new name and then clicking the **Rename** button.

Rename — Clicking the **Rename** button overwrites the selected macro name in the **Names** list box with the name you entered in the **Rename** field.

Remove — The **Remove** button will remove any selected entry from one of the three list boxes.

File — The **File** button will write all currently defined macros to files in the **Scripts** folder of the ***DraftBoard*** directory. The file names are generated directly from the macro names. The files are output in such a way that when ***DraftBoard*** is subsequently started, the macros will be activated automatically. **Macro** files can be moved to other locations, or to other machines for use in other copies of ***DraftBoard***.

Run — The **Run** button will run the macro currently selected in the **Names** list box. Macros can call other macros if

they are in the **Utilities** menu. This is done by clicking on the relevant entry in the **Utilities** menu, like any other menu item. This feature allows a macro to call another macro, which in turn could call the original macro. Such *Infinite Loops* are checked when an attempt is made to include a macro, and if detected, an alert box is posted resulting in the macro not being included.

Names list box — In this list box all names of the macros defined are displayed alphabetically. When one of the items is selected with the cursor, the **Strokes** and the **Commands** list boxes, as well as the **Key** and **In Utilities Menu** fields are automatically filled in with the relevant data that defines the selected macro.

Strokes list box — The **Strokes** list box shows a group of stroke commands, if any, that can run the selected macro. The names in this list box relate to the encoding of the shape of the stroke and you need not be concerned about them. It is optional to associate a stroke to define a macro.

Commands list box — The **Commands** list box shows the set of menu that define this macro. They are shown in this form:
MenuName; **EntryName**.

Key — You can assign your own key combinations for a macro by clicking in the **Key** edit box and entering the key combination or typing it directly in the **Key** entry field. For example, enter **Ctrl** and **F1** (by pressing the **Control** key first, keeping it depressed and pressing then the key **F1**). The **Key** edit box will display **Ctrl F!**. If you try to create a **Control** key combination, you will not be able to execute the macro if the key combination is already in use.

In Utilities — If you mark this option the macro name will be included as a menu item in the **Utilities** menu to access the macro directly.

Creating a macro

1. Select the **Macro** command in the **Utilities** menu.
   The **Macro** dialog box is displayed
2. Click the **New** Button.
   The name **Macro1** is displayed in the **Rename** field.
3. Overwrite the name if desired.
4. Select menu commands in the order you want the new macro to execute.
   The selected commands are automatically filled in the **Commands** list box.
5. Define a **Stroke** command on the drawing area.
   Depress the **Control** and **Shift** keys (*Macintosh:* ***Command key***) and ***drag*** a symbol on the drawing area you want to be associated to the new macro.
   That can be letters or a figure like a circle. You can not use ***DraftBoard's*** *Standard Stroke* commands like *Zoom* or *Construction* strokes, since they would be executed immediately. A letter and number combination describing the executed **Stroke** will be filled into the **Stroke** list box automatically. When you define a stroke, you must repeat it several times to define all variations of the stroke, so that all possible variations are covered and can be recognized by ***DraftBoard***.
6. Mark the **In Utilities Menu** option if you want the new macro included in the **Utilities** menu.
7. Enter a **Key** short cut in the **Key** entry field.
   Click in the **Key** entry box and enter the key combination or type it directly in the **Key** entry field.
8. Close the **Macro** dialog box by clicking the **Close** box in the upper left corner of the dialog box.

Editing a Macro

1. Display the **Macro** dialog box.
2. Select a **Macro** in the **Names** list box.
3. Rename the **Macro** in the **Rename** entry box and click the **Rename** button.
4. Redefine the **Macro** by executing menu items and assigning a new stroke equivalent to the macro.
5. Alter the **Macro** key combination in the **Key** entry box.
6. Close the **Macro** dialog box by clicking the **Close** box in the upper left corner of the dialog box.

Removing a Macro

1. Display the **Macro** dialog box.
2. Select a **Macro** in the **Names** list box.
3. Click the **Remove** button.

Running a Macro

Each **Macro** has a unique name and can be run in one of four ways:

1. It can be invoked with a **Control** key combination.
   Subsequent typing of that **Control** key combination will cause the associated macro to run.
2. A macro can be invoked by a family of stroke commands you defined in the **Macro** dialog box.
3. You can select the macro in the **Extras** menu if you marked the option **In Extras Menu** in the **Macro** dialog box.
4. You can display the **Macro** dialog box, select a macro in the **Names** list box and click the **Run** button.
   Executing a macro with the **Run** button is useful for testing a macro. Should it not work properly, you can highlight any dialog box entry and remove it by pressing the **Remove** button. Modify your macro as needed then test it again by pressing the **Run** button. Continue this process until your macro runs properly.

Macro Limitations

Macros are not general programs. There are limitations as to what can be encoded in a macro. The following rules apply to macros.

- Only menu items can be included in a macro.
- A menu item might cause a dialog box to appear, but the macro cannot fill in the required entries.
- There are no conditionals (like **if** or **ifelse**) or loops (like **while**).
- Subroutines are allowed in that one macro can call another macro, provided the called macro is in the **Extras** menu and therefore a menu item. If a macro calls a nested macro, which would end in an *infinite loop,* an alert box will be posted and the macro will be excluded from execution.

Function Interface

The *Function interface* allows you to add *functions* and *commands* to ***DraftBoard***. These functions can be displayed in an own palette and the added commands will be displayed in the **Utilities** menu.

DraftBoard is shipping already with the following additional functions:

- Automatic Dimensioning
- Dimension axes
- Text Lines

and the following commands:

- Check Dimensions
- Remove dots

With the function interface you can add easily functions and commands that were either especially programmed for you or downloaded from the ***DraftBoard*** *Web Page* (**www.draftborad.de**).

All commands for adding functions and commands to ***DraftBoard*** are listed in the **Extra** menu.

All functions and commands you want to bind into ***DraftBoard*** must be in the **Function** directory or folder of ***DraftBoard***.

Reload

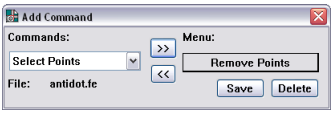
This commands allows you to reload additional *functions* and *commands* while ***DraftBoard*** is already running, since only loaded functions and commands can be added into the **Utilities** menu and displayed in the additional function palette.

All *functions* and *commands* that are placed in the **Function** folder are loaded automatically when ***DraftBoard*** is launched.

If you copy additional *functions* and *commands* into the **Function** directory while ***DraftBoard*** is already running they must be reloaded using the **Reload** command in the **Extra** menu.

Add Commands

When you call this command the following dialog box is displayed:



The **Add Commands** dialog box allows the following settings:

Commands — This list box contains all commands from the **Function** folder that can be added into the **Utilities** menu. These commands were automatically loaded when ***DraftBoard*** was started or reloaded using the **Reload** command when ***DraftBoard*** was already running.

File — This field shows the *file name* of the source code file for the selected command in the **Commands** list box.

Arrows — Clicking this buttons moves a command from the **Commands** list box to the **Menu** list box and vice versa.

Menu — This list box contains all commands that will be added as new menu items.

Clear — Using this commands moves all commands from the **Menu** list box to the **Commands** list box and removes all added commands from the **Extra** menu.

Save — Clicking the **Save** button adds all commands contained in the **Menu** list box into the last section of **Utilities** menu.

All added commands are saved permanently and are available each time you launch ***DraftBoard***. To remove these commands from the **Extra** menu you must first remove the commands from the **Menu** list box using the **Clear** button and click then the **Save** button again.

Adding commands

1. Copy the desired *Source code files* into the **Function** directory of ***DraftBoard***. *Source code* files containing *commands* and *functions* have the file extension **.fe**.
2. Select the **Reload** command in the **Extra** menu if you have copied the *source code files* in to the **Function** folder when ***DraftBoard*** was already running.
3. Select **Add Commands** in the **Extra** menu.
   The **Add Commands** dialog box is displayed.
4. Move all commands you want to add into the **Extra** menu from the **Commands** list box to the **Menu** list box using the **Arrow** buttons.

Commands you want to remove from the **Utilities** menu you must move from the **Menu item** list box to the **Commands** list box using the **Arrow** buttons showing in the opposite direction.

5. Click the **Save** button.
   All commands contained in the **Menu** list box are added into the last section of the **Extra** menu.

Remove single commands

When you delete a source code file in the **Function** folder of the **DraftBoard** folder, the related command is automatically removed from the **Extra** menu when ***DraftBoard*** is launched again.

1. Select the **Add Commands** in the **Utilities** menu.
   The **Add Commands** dialog box is displayed.
2. Select the command in the **Menu** list box you want to remove from the **Extra** menu.
3. Click the appropriate **Arrow** button.
   The selected commands are moved from the **Menu** list box to the **Commands** list box.
4. Click the **Save** button
   All selected commands are removed from the **Utilities** menu.

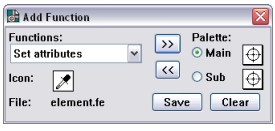
Remove all commands

1. Select the **Add Commands** in the **Utilities** menu.
   The **Add Commands** dialog box is displayed.
2. Click the **Clear** button.
   All commands are moved from the **Menu** list box to the **Commands** list box.
3. Click the **Save** button
   All commands are removed from the **Utilities** menu.

Add functions

This command allows you to add additional functions into a separate function palette.

When you call the **Add Functions** command the following dialog box is displayed:



The **Add Function** dialog box allows the following settings:

Functions — The **Function** list box contains all function from the **Function** folder that can be added to the additional *function palette*. These functions were automatically loaded when ***DraftBoard*** was launched or reloaded using the **Reload** command when ***DraftBoard*** was already running.

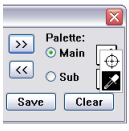
File — This field shows the *file name* of the source code file for the function selected in the **Functions** list box.

Icon — shows the icon of the function selected in the **Functions** list box.

Arrows — These buttons move functions form the **Functions** list box to the two palettes and vice versa.

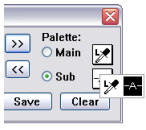
Palette — The two radio buttons **Main** for *Main palette* and **Sub** for *subpalettes* allow to configure a function palette that is handled like the *Standard Function Palette* of ***DraftBoard*** .

Main — Clicking the **Arrow** button while this radio button is marked moves the selected function from the **Function** list box to the *Function palette* and displays the related icon in the palette. Using this option all functions will be placed vertically without building any subpalettes.



The palette shows always the icon of the last function that was moved to the palette. You can display all function that are contained in the palette by clicking the icon and keeping the mouse button pressed. As long the mouse button is pressed all icons contained in the palette are displayed.

Sub Clicking the **Arrow** button while the **Sub** radio button is marked moves the selected function from the **Function** list box to the *Function palette* and displays the related icon in the palette. Using this option all functions will be placed horizontally creating a sub-palette for the icon displayed in the palette.



The palette shows always the icon of the last function that was moved to the palette. You can display all function that are contained in the subpalette by clicking the icon and keeping the mouse button pressed. As long the mouse button is pressed all icons contained in the subpalette are displayed.

Clear Using this commands clears the palette and moves all commands from the **Palette** to the **Functions** list box.

Save Clicking the **Save** saves the generated function palette that can be displayed with the **Show Palette** command in the **Extra** menu.

The new function palette is saved permanently and is available each time you launch ***DraftBoard***.

Creating an additional Function palette

1. Copy the desired *Source code files* into the **Function** folder in the **DraftBoard** folder. *Source code* files that contain *commands* and *functions* have the file extension **.fe**.
2. Select the **Reload** command in the **Extra** menu if you have copied the *source code files* in to the **Function** folder while ***DraftBoard was*** already running.
3. Select the **Add Functions** command in the **Extra** menu.
   The **Add Functions** dialog box is displayed.
4. Move all functions you want to place in the **palette** from the **Functions** list box to the **Main** or **Sub** palettes using the **Arrow** buttons.
   Using the **Arrow** buttons showing in the opposite direction you will move functions from a **Palette** to the **Functions** list box.
5. Click the **Save** button.

The function icon is always copied as well to the sub-palette since it is as well the first icon of a possible subpalette.

Removing a single Function

1. Select the **Add Functions** command in the **Extra** menu.
   The **Add Functions** dialog box is displayed.
2. Select an icon of a function in one of the palettes you want to remove from the palette.
3. Click the appropriate **Arrow** button.
   The selected function is moved from the **Palette** list to the **Functions** list.
4. Click the **Save** button.

When you delete a source code file in the **Function** folder of the **DraftBoard** directory the related function is automatically removed from the *Function palette* when ***DraftBoard*** is launched again.

Removing all Functions

1. Select the **Add Functions** command in the **Utilities** menu.
   The **Add Functions** dialog box is displayed.
2. Click the **Clear** button.
   All functions are moved from the palettes to the **Functions** list.
3. Click the **Save** button
   All functions are removed from the palettes.

Show Palette

This command displays the palette you have generated with the **Add functions** command in the **Extra** menu. When no palette is available this command displays only the **Title bar** of the palette.

To remove the palette you select either the **Hide Palette** command in the **Extra** menu or click the **Close** box of the palette.

Examples for additional Functions and Commands

DraftBoard is already shipping with some functions and examples that you can bind into ***DraftBoard***.

These functions are not officially released and don't go through the same quality control as ***DraftBoard***. Further functions can be created with the ***DraftBoard programming*** language **FE** (**F**ront **E**nd) or downloaded from the ***DraftBoard*** *Web Page* (**www.draftboard.de**). Call your local ***DraftBoard*** distributor for more detailed information.

The documentation of the functions do not describe necessarily the current version of a *function* or *command*. Since their functionality is permanently extended there could already exist an improved version of these tools and commands. Read therefore always the related readme file of the related function in the **Function** folder of the **DraftBoard** folder before you bind in a function or command into ***DraftBoard***.

Remove Dots

Before you can use this command you must add it to the **Utilities** menu using the **Add Command** command in the same menu. (See as well under *Adding Commands* in this chapter).

This command removes all isolated construction points from the drawing that were created for example by erroneous mouse clicks.

Removing dots

1. Choose the **Remove Dots** command from the Extra menu.

   All isolated construction points are removed from your drawing.

Text Lines

This functions creates horizontal *Text lines*.

Creating Text lines

1. Draw a horizontal line.
2. Select **Show Palette** in the **Extra** menu.
3. Mark the line where you want to insert text using the **Selection** function in the standard Function palette.
4. Select the **Text Line** function.
5. Enter in the status line the desired text.
6. Press the **Enter** key.

   The text is inserted into the middle of all selected horizontal lines. The line segment behind the text is deleted. You can edit the text using the **Text** function in the standard Function palette.

   If you want to move the Text within the line you have selected with a selection marquee the text and the endpoints of the adjacent lines.

When you want to move the inserted text along the line you must select the text *including* the neighbouring line endpoints with a selection marque using the **Selection** function in the standard ***DraftBoard*** tool palette

Automatic Dimensioning

This function dimensions automatically *Lines* independently from their angle, *Circles*, *Ellipses* and *arcs*.

Using Automatic Dimension

1. Draw a circle, several lines in different angles and an arc.
2. Select **Show Palette** in the **Utilities** menu.
3. Choose the **Automatic Dimensioning** function.
4. Click at each object with the **Automatic Dimensioning** function.

   All *lines* are dimensioned correctly, all *circles*, *arcs* and *ellipses* are dimension with a *radius* dimension.

Specifying Crosshatch Patterns

DraftBoard offers a wide variety of *DIN* and *ANSI hatch patterns* with editable pattern *spacing* and *angle*. You will find these hatches organized in groups such as *metals*, *fluids* etc. in the **Hatch** dialog box.

Adding and specifying Hatch Patterns

In addition to the hatch patterns shipping with ***DraftBoard*** you can use all hatch patterns that are compatible with the hatch format **.pat** of AutoCAD©. When you copy these hatch files into the **DraftBoard** folder **DraftBoard\Scripts\VSG\Hatches** the file name is automatically displayed in the **Crosshatch** dialog box. If the file contains more than one hatch pattern the name is followed by an ellipsis **(...)**. When you double-click the file name all patterns contained in the file are displayed.

How to define your own hatch patterns is described under *Hatches* in the *Appendix* of this manual.

Adding Plotter Fonts

In addition to the plotter fonts shipping with ***DraftBoard*** you can use all plotter fonts that are compatible with the file format **.shx** of AutoCAD©. When you copy these files into the **Fonts** folder of the **DraftBoard** folder, the new fonts are displayed in the **Fonts** submenu after relaunching ***DraftBoard***.

When you want to print ***TrueType*** *Fonts* on pen plotters you must use a ***WinLINE Plus*** *Plotter Driver*. Look for more information about ***WinLINE*** *Plus* in the Internet under ***www.winline.com***.

CHAPTER

5

Selecting Objects

Objects

Indicating Selection

Selection Process

Selecting Objects

Once you have constructed objects, you may want to make changes. One of the basic rules of ***DraftBoard*** says that you must first select an object before you can edit it. Just as you first have to select a drawing tool from the **Tool palette** before you can start drawing, an object must first be selected before a function can be applied.

Modifying an object is always a two-step process:

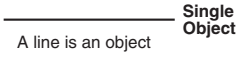
- Select the object.
- Specify the action for the selected object.

For example, you could select a circle and then change the pen style to **Center** to indicate a bolt-hole circle.

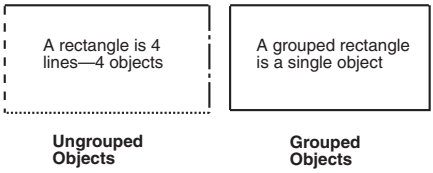
This chapter deals with selecting both *objects* and *points*, since they behave in different ways.

Objects

A single piece of geometry (for example a point or a line) is an object.

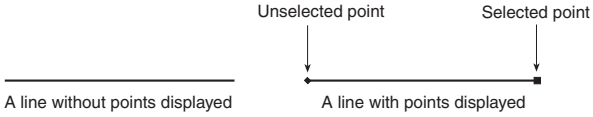


Several objects that have been grouped with the **Group** command are also an object. For example, the four lines of a *Single Line* rectangle are four objects. If you group them, ***DraftBoard*** treats them as a single object.



A point is an object, too. Every type of geometry contains two or more points, sometimes called *control points*. A line has two control points, one at the beginning and one at the end of the line. If you select an object and choose **Show Points** from the **Layout** menu, the points are visible on the selected object.

The *Starting* and *Ending point* of a circle is displayed by the *Drawing Assistant* as ***Vertex***.



If you select a point without selecting the geometry it defines, ***DraftBoard*** treats the point as an object.

Point selection is described in a section later in this chapter.

Note*: Keep in mind you need to select an object and than the function.*

Indicating Selection

When you select an object, its appearance on the screen shows that it is selected, it is a specific color (such as red), it blinks, or both. ***DraftBoard*** usually displays selections according to your monitor type, red for a color monitor and blinking for a monochrome monitor. You can change the color or turn blinking off and on for the selection indicator. You can choose both color and blinking, if you want.

Selection

This command in the **Preferences** submenu in the **Layout** menu determines the appearance of selected objects. If you have a color monitor, selected objects appear in a **Color** (which you can change with this setting). If you have a monochrome monitor, selected objects **blink**.

If you have a color monitor, you can also specify blinking with or without color.

Indicating the Selection

5 You shouldn't use black for indicating selection because many of the standard pens use black lines. Red is a good choice unless you are color blind, in which case blue may be a better choice.

1. Choose the **Selection** command from the **Preferences** submenu in the **Layout** menu.

   The **Selection** dialog box is displayed.

2. Select the option **Blinking** and/or **Color**.

   To choose a color, move the pointer to the color box and click into the box to display the choices. ***Drag*** to the color you want and release or click the color.

   It is not recommend to use the color **Black** to indicate the selection, since many lines are displayed in black.

3. Click the **OK** button.

   Once the selection color is changed, all current and future selected items appear in the new color until you quit ***DraftBoard***.

   Using the **Save Preferences** command you can save the new settings permanently.

Selection Process

Selecting an object does not affect the properties of the object. A selected object is highlighted, but this highlighting goes away once the object is deselected. While points and objects are selected in a similar way, point selection is controlled by the **Selectable Points** setting in the **Edit** menu. For this reason, selecting objects and selecting points are discussed separately.

Selection Tools

The **Selection** tool palette contains all tools for selecting objects.

Objects can be selected by *clicking*, by ***dragging*** a *Selection Marque* around a group of objects, by ***dragging*** a *Line* over objects or by drawing a *Selection Polygon* around objects. In the **Selection** tool palette you find as well the **Tracer** tool for automatically selecting boundary lines.

All **Selection** tools allow when pressing the **Shift** key to select or deselect additional objects.

Selection Tool

The **Selection** tool on the tool palette selects one or more objects or points in the drawing area.

Selecting a single object

1. Click the **Selection** tool in the tool palette.

2. Move the pointer to an object and click.
   The object is selected, and previously selected objects are deselected.

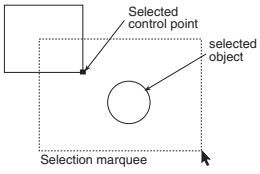
Selecting multiple objects by shift-clicking

1. Click the **Selection** tool in the tool palette.
2. Move the pointer to an object and click.
3. Press and hold down the **Shift** key.
4. While holding down the **Shift** key, click other objects to be selected.
   The objects you clicked are selected. If you then click one of the selected objects, it is deselected.
5. Release the **Shift** key.

Selecting multiple objects by *dragging*

If you want to select more than one object, you can ***drag*** with the mouse button pressed a selection marquee around the objects.

1. Click the **Selection** tool in the tool palette.
2. ***Drag*** a selection marquee around the objects you want to select.



   All objects that lie completely inside the selection marquee are selected. If a portion of an object is outside the region you ***dragged***, that geometry is not selected but the *control points* of that geometry which lie inside the selection marquee are selected.

If you want to select most of the objects within an area, you can ***drag*** a selection marquee to select all the objects, and then deselect the objects you do not want selected by holding down the **Shift** key and clicking them.

Select all Objects

Double-clicking the **Selection** tool selects all objects on the drawing area.

Selecting an object that is overlapped by another object MAC

MAC

Occasionally, you may have geometry that overlaps other geometry. To select geometry underlying the object, follow these steps:

1. Click the **Selection** tool in the tool palette.
2. Select the geometry on top.
3. Hold down the **Shift** key.
4. Hold down the **Ctrl** key and click the selected geometry again.
   The obscured geometry is selected.
5. Release the **Shift** and the **Ctrl** key.

Tracer

This tool on the **Selection tool** subpalette selects automatically boundary lines. The tool uses coincident endpoints and line intersections to trace the perimeter of the geometry.

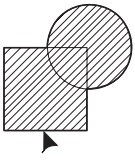
The **Tracer tool** is most useful for selecting boundaries for *crosshatching*, for *solid fills* and for *2D Analysis* because it eliminates the need to segment geometry.

Selecting boundary lines

1. Select the **Tracer tool** from the **Selection Tool** subpalette.
2. Click on a boundary line.
   The tracer moves from one line to another making its selection.

*Important: You can press the **Escape** key to stop the **Tracer**.*

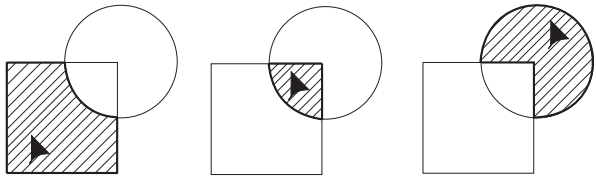
If you have two overlapping objects as shown on next page, the tracer selects the perimeter of the objects, depending on where you click.



*Important: The **Tracer** tool does not work on a 3D model.*

If you click on the outside of the perimeter line, the **Tracer** selects the outer most perimeter. If you click on the inside of the line, the **Tracer** selects the inner most perimeter.

The crosshatching on the figures below shows the boundaries selected when you click the **Tracer** tool at the cursor locations shown.



Intersecting areas

If you want to determine and isolate the overlapping part of two intersecting areas you proceed as follows:

The **Tracer tool** creates a *temporary* copy of the boundaries. Temporary because it's copied to a *temporary* layer, which is deleted as soon you activate the **Tracer tool** or select a command other than **Hatch** or **2D-Analysis**. As long as the *temporary* boundaries are selected they can be copied to another layer.

1. Select the **Tracer tool** from the **Selection Tool** subpalette.
2. Click on the overlapping area.
   The overlapping area is selected.
3. Select **Layer** in the **Layout** menu.
4. Create a new layer.
5. Select **Edit Objects** in the **Layout** menu.
6. Select the new layer in the **Layer** list box and click the **Apply** button
7. Close the **Edit Objects** dialog box.
8. Make the new layer in the **Layer** dialog box the work layer.
9. Hide the layer that contains the overlapping object geometry.

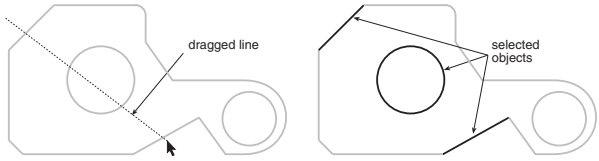
The new layer contains now the intersecting area which you can edit like any other object.

Select by Line

This tool selects one or more objects along a straight line path when you ***drag*** a line over the desired objects.

Selecting an object by Line

1. Choose the **Select by Line** tool from the **Selection** Tool Palette.
2. ***Drag*** a line across the objects you want to select.



The objects intersected by the line are selected and any previously selected objects are deselected.

You can select/deselect additional objects by pressing the **Shift** key. You can move a copy of the last selection by holding down the **Ctrl** key (*Macintosh:* ***Option*** *key*).

Select by Polygon

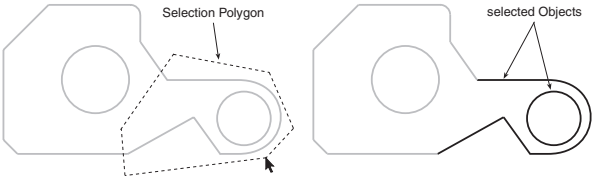
This tool selects one or more objects when a polygon is drawn around them. Each object must be completely enclosed by the selection polygon for the object to be selected. This tool functions similarly to the **Connected Lines** tool.

Selecting objects by Polygon

1. Choose the **Select by Polygon** tool in the **Selection** Tool palette.
2. Click or ***drag*** to indicate the endpoints of the line segments that will make up this selection polygon. The selection polygon appears as line segments and are selected (the default is red).

   If you click a point and then change your mind, press the **Escape** key or choose **Undo** to remove the last line. Pressing the **Backspace** key (*Mac:* ***Delete***) removes the entire selection polygon.

2. Indicate the last point by double-clicking the startpoint of the polygon or choosing another tool.



Objects completely within the selection polygon will be selected. Objects not completely within the selection polygon will not be selected.

When pressing the **Shift** key you can select/deselect additional objects by additional polygons. It is not possible to move a copy of the last selection with this tool.

Selection commands

The **Edit** menu contains the two selection commands **Select All** and **Selection Mask**.

Select All Ctrl+A [Macintosh: ⌘A]

This command in the **Edit** menu selects all objects except those on a hidden layer or excluded by the **Selection** mask. You can also double-click the **Selection** tool to select everything or use **Ctrl+A** (*Macintosh:* ***Command key*** *+**A***) to select all objects. If you choose **Select All** while using the **Text** tool, all characters in the current text area are selected.

Select All is useful if you want to make a global change in a drawing, such as changing the width of all lines. If you want to specify that only certain objects, layers, or colors are selected you should use the **Selection Mask** command in the **Edit** menu.

Selection Mask

This command in the **Edit** menu allows you to limit selection by *object type*, *layer*, and *color*. You can only select objects that are highlighted in the dialog box.

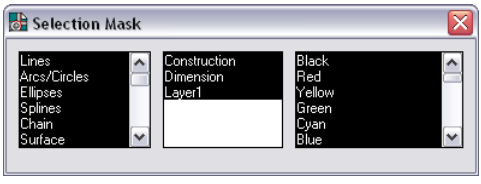
For example, if circles are not highlighted, when you choose **Select All** from the **Edit** menu, everything but the circles is selected. In this way, you can select such combinations as only blue splines or only red objects on a particular layer.

The selection mask is useful when you have created a complex drawing and want to change particular groups of objects. You can use this feature to export some but not all geometry.

Using the selection mask

1. Choose **Selection Mask** from the **Edit** menu.

   The **Selection Mask** dialog box appears.



   The highlighted items respond to all selection methods and can be detected by the Drawing Assistant.

2. Click the items you want to be able to select so they are highlighted.

To deselect a list quickly, click one item in the list.

You can select a contiguous group by ***dragging*** or holding down the **Shift** key and clicking.

If you want to select or deselect non-contiguous items, hold down the **Ctrl** key (***Macintosh***: ⌘-key) and click the items.Using *Windows* you select all items in a list by double-clicking one item in the list. On the *Macintosh* you select all items in a list by pressing the **Command** key and double-clicking one item in the list that is not selected.

While the dialog box is visible, you can select, create, and edit geometry. You can move the dialog box if it covers geometry you want to work on. The selection mask remains in effect even after you close the dialog box until you select a tool.

When an item in the dialog box is not highlighted, the *Drawing Assistant* and all the tools cannot detect it, even though it is visible to you on the screen.

Deselecting

To deselect an object, click anywhere in the drawing area where there is no object, or click any tool in the tool palette.

You can deselect an object that was selected in a multiple selection operation. While the objects are still selected, hold down the **Shift** key and click the objects you want deselected.

Selecting Points

Selecting points differs from selecting objects because points are not always visible.

Being able to select points is useful in two situations:

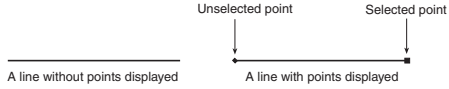
- stretching selected geometry (described later in this chapter)
- control point selection for transformations.

When you ***drag*** a selection marquee around objects, all geometry that is completely within the marquee is selected. If geometry is partially within the marquee, only the control points inside the marquee are selected, and the geometry is not selected.

Show Points

This command in the **Layout** menu toggles the display of the *control points* (endpoints, midpoints, center points, and knot points) for selected objects. When points are displayed, you can select a point by clicking it. If points are not displayed (but **Selectable Points** is set in the **Edit** menu), you can select a point by ***dragging*** a selection marquee around the location of the point.

To show points for an individual selection, use the **Edit Objects** command in the **Edit** menu.



Displaying points

1. Select the geometry.
2. Choose **Show/Hide Points** from the **Layout** menu to toggle the display of points *on* and *off*.

To turn off the point display once the points for an object are showing, you must select the object again and choose either **Show/Hide Points** from the **Layout** menu or the **Control Points** option in the **Edit Objects** dialog box.

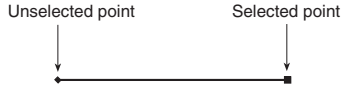
Showing and Hiding Points with Stroke

When you hold down the **Shift** and **Ctrl** key and click an object, the display of the object's points toggles on or off. If the points are hidden when you click the object, the points will be displayed.

Selecting invisible points

You can select a control point whether the points are visible or not; however, note that **Selectable Points** in the **Edit** menu must be on (displaying the ✓ symbol in the menu). To select points if points are not visible:

1. Click the **Selection** tool.
2. ***Drag*** a selection marquee around the location of the point.
   The selected point displays as a square.



Selecting visible points

To select points if they are visible:

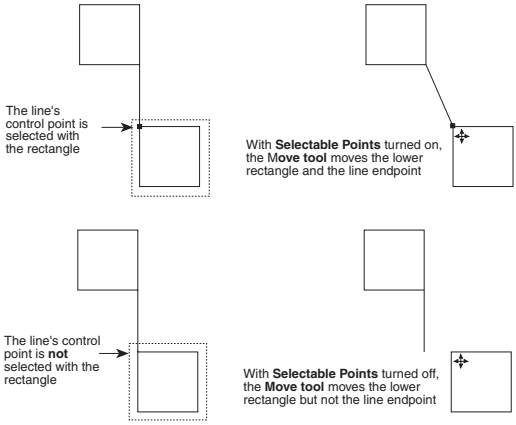
1. Click the **Selection** tool.
2. Click the point.
   The selected point displays as a square.

Selectable Points

This command in the **Edit** menu gives you the ability to select points that aren't displayed. When **Selectable Points** is not set, you cannot select points by ***dragging*** a marquee.

If points are displayed, simply click the point to select it. If points are not displayed, you can select a point by ***dragging*** a selection marquee around it.

The following example illustrates the use of **Selectable Points**.


The line's control point is selected with the rectangle
With **Selectable Points** turned on, the **Move tool** moves the lower rectangle and the line endpoint
The line's control point is **not** selected with the rectangle
With **Selectable Points** turned off, the **Move tool** moves the lower rectangle but not the line endpoint

If you select and ***drag*** a line, the line and the endpoints move. If you select and ***drag*** only an endpoint of the line, the endpoint moves and the line length changes while the other endpoint of the line remains fixed.

CHAPTER

6

Editing Objects

Editing Tools

Moving Objects with Tools

Copying Objects with Tools

Sizing Objects with Tools

Editing Commands

Duplicating Objects

Changing the Characteristics of Objects

Arranging Geometry

Editing Objects

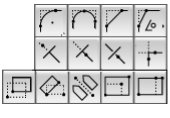
Once you create an object, you can select and alter it by choosing a command or selecting a tool to edit it. This section describes the common editing activities you can perform on a selected object.

The chapter also compares the different methods for performing the same action, such as the difference between ***dragging*** a selected item and using the **Move** tool.

All procedures in this chapter are described for single objects, they naturally are valid as well for several selected objects.

Editing Tools

These tools allow you to change an object physically, either altering the geometry, changing the size, or changing the location or orientation of the selected object. Three subpalettes on the tool palette provide these editing capabilities.



Fillet and Chamfer Tools
Trim Tools
Transformation Tools

Fillet and Chamfer	The *Fillet and Chamfer* tools round the corners of nonparallel lines or cut them at straight angles, respectively.
Trim	The *Trim* tools cut off or extend a line to the boundary limit you specify. You can also segment a line at an intersection, or create corners from intersecting lines.
Transformation	The *Transformation* tools *move*, *rotate*, *expand* or *shrink*, *stretch* and *mirror* objects.

Fillet and Chamfer Tools

These tools on the tool palette construct fillets and chamfers from corners formed by nonparallel lines or curves. The fillets and chamfers are automatically trimmed. If you do not want them trimmed, hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while you are selecting the objects to fillet or chamfer.

2-Entity Fillet Tool

Before you can fillet *Smart walls* you have to ungroup them.

This tool constructs an arc tangent to the two objects you click.

Adding a 2-Entity Fillet

1. Click the **2-Entity Fillet** tool.
2. Enter the arc radius you want in the status line. The default radius is **5 mm.**
3. Click the objects you want to fillet. You can also hold down the **Shift** key and click once inside the corner you want to fillet.

When filleting two parallel lines the fillet will be tangent to all objects only if the filleting radius is half the distance between the two parallel lines.

If you hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while you select the objects to fillet, the objects are not trimmed.

Fillets use the smallest arc between the selected geometry.

The status line allows you to specify the **R**adius of the fillet either before or after you create the fillet.

3-Entity Fillet Tool

This tool constructs a fillet tangent to the three objects you choose.

Adding a 3-Entity Fillet

1. Click the **3-Entity Fillet** tool.
2. Click the three objects you want to fillet.

When you fillet two parallel lines the filleting radius has to be half the distance between the two parallel lines.

If you hold down the **Ctrl** key (*Macintosh*: **Option** key) while you select the objects to fillet, the objects are not trimmed.

There are no entries on the status line.

2-Entity Chamfer Tool

This tool creates a chamfer across a corner at the specified distance from the intersection of two lines. The default distance is **5** mm.

Adding a chamfer

1. Click the **2-Entity Chamfer** tool.
2. In the status line, enter the distance you want the chamfer from the corner. The default distance is **5** mm.
3. Click each line making up the corner you want chamfered. You can also hold down the **Shift** key and click once inside the corner you want to chamfer.

The lines are automatically trimmed or extended. If you hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while you select the objects to chamfer, the objects are not trimmed.

The status line allows you to specify the distance (**L**ength) from the chamfer to the intersection of the corner lines.

Angular Chamfer

This tool creates a chamfer at the specified angle and distance from the corner. The specified angle is the angle between the chamfer and the second line of the corner. The specified length is the distance between the corner and the intersection of the chamfer and the second line of the corner. The default distance is **0.5 mm** and the default angle is **45°**.

Adding a chamfer

1. Click the **Angular Chamfer** tool.
2. In the status line, enter the **length** you want the intersection of the chamfer and the second line of the corner from the corner. The default length is **0.5 mm**.
3. In the status line, enter the angle you want between the chamfer

and the second side. The default angle is **45°**.

4. Click each line making up the corner you want chamfered. You can also hold down the **Shift** key and click once inside the corner you want to chamfer.

The lines are automatically trimmed or extended. If you hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while you select the objects to chamfer, the objects are not trimmed.

The status line allows you to set the distance (**L**ength) from the intersection as well as the angle.

Trim Tools

The *Trim tools* on the tool palette lengthen or shorten lines and curves. In general, you select the limiting object(s) before you select the tool. However, if you don't select everything you need, you can hold down the **Shift** key and click additional objects after the *Trim tool* is selected.

Simple Trim Tool

This tool shortens a line to the specified boundary. If you hold down the **Ctrl** key (*Macintosh*: **Option** key) while using this tool, it becomes the **Relimit** tool.

Rule: Point to what you want to throw away.

Trimming objects

1. Select the object that limits the trim.
2. Select the **Trim** tool.
   If necessary, use **Shift**-Click to select more boundary objects.
3. Click the section of the object to be discarded.

There are no status line entries.

Relimit Tool

This tool lengthens or shortens a line to the specified boundary. If you hold down the **Ctrl** key (*Macintosh*: **Option** key) while using this tool, it becomes the **Trim** tool.

Rule: Point to what you want to keep.

Relimiting objects

1. Select the object that limits the change.
2. Select the **Relimit** tool.
   If necessary, use **Shift**-Click to select more boundary objects.
3. Click the section of the object that you want to remain.

There are no status line entries.

Segment Tool

This tool divides a line or curve at intersections with other lines or curves.

Segmenting an object

1. Select the objects that limit the segmentation.
2. Select the **Segment** tool.
   If necessary, use **Shift**-Click to select more boundary objects.
3. Click the object to be segmented.

The selected object is segmented at the boundary objects. Even though you can not see the segmentation on the screen, you can select parts of the segmented line by clicking.

If you hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while selecting the line to be segmented, the new segment appears in the characteristics of the *current pen* style.

There are no status line entries.

In order to see the segmentation on the screen, first select the boundaries and the object you want to segment and then **Show points** in the **Layout** menu. After the segmentation, the endpoints of the segmented parts are displayed.

Corner Trim Tool

This tool creates a corner from the two objects you specify. Lines are extended or shortened to create the corner.

Creating a corner

1. Click the **Corner Trim** tool.
2. Click each object. You can also press and hold the **Shift** key and click inside the about-to-be-created corner.

There are no status line entries.

Extending lines to a theoretical intersection

If you want to extend a line to its theoretical intersection with another line, first click the line to be extended, then hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*) and click the line that is not to be trimmed.

Transformation Tools

These tools on the tool palette *move*, *rotate*, *expand* or *shrink*, *stretch* and *mirror* objects. Select the object you want to transform before you select the *Transformation tool*.

You can copy at the same time you transform objects by holding down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while you specify the transformation.

When you press the **Shift** key you can select additional objects after you have selected a Transformation tool. As soon as you release the **Shift** key the transformation tool is active again.

Move Tool

How to enlarge or shrink an object with the **Move** tool is described under *Sizing objects with Tools* in this chapter.

This tool moves the selected objects to a new location. You can copy the selection by holding down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while you select the objects. If you select more than one object, they remain in the same position relative to each other.

Moving objects

A positive or negative value entered in the status line determines the direction along the **X** or **Y** axis. A negative value moves the object to the left or down on the screen. A positive value to the right or up.

1. Select the object(s) to be moved.
2. Select the **Move** tool.
   If necessary, use **Shift**-Click to select more objects.
3. ***Drag*** the selected object to a new location, pressing to set a reference point and releasing to indicate the new location.

You can also click a reference point, then click a destination point to move the selected object to the new location. It is not necessary for the reference and destination points to be on the object you are moving.

The move is performed relative to the specified points.

The status line allows you to specify the **X**,**Y** distance that the selection is to be moved.

Rotate Tool

This tool rotates one or more objects around a specified point. You can copy the selection by holding down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while you select the objects. If you select more than one object, they remain in the same position relative to each other.

Rotating objects

1. Select the object(s) to be rotated.
2. Select the **Rotate** tool.
   If necessary, use **Shift**-Click to select more objects.
3. Specify the **center** of rotation.

4. ***Drag*** the objects or enter an **angle** in the status line to specify the rotation.

You can also click the *reference* point and then click the *destination* point. It is not necessary for the reference and destination points to be on the object you are moving. If they are not, the rotation is performed relative to the specified points.

The status line allows you to specify the **A**ngle of rotation.

Mirror Tool

This tool creates the mirror image of an object or objects on the opposite side of a reference line. You can copy the selection by holding down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while you select the objects. If you select more than one object, they remain in the same position relative to each other.

1. Select the object(s) to be mirrored.
2. Select the **Mirror** tool.
   If necessary, use **Shift**-Click to select more objects.
3. Specify the *reference line* by clicking two locations or by ***dragging***. The *reference line* need not be parallel to the object.

If the mirror axis should be aligned in a precise angle, you have to create a construction line before.

The status line has no entries.

Expand/Shrink Tool

This tool enlarges or reduces objects while maintaining the proportions. You can copy the selection by holding down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while you select the objects. If you select more than one object, they remain in the same position relative to each other.

Using the beginning and ending reference point for the **Expand/Shrink** tool is particularly useful if you want to change the size of an object relative to the size of another object.

Expanding or shrinking objects

1. Select the object(s) to be expanded or shrunk.
2. Select the **Expand/Shrink** tool.
   If necessary, use **Shift**-Click to select more objects.
3. Click a point on the object that you want to remain stationary.
4. ***Drag*** the selected object to a new location, pressing to set a reference point and releasing to indicate the new location.

When you want to change the size of an object relative to the size of an other object you should click the *Reference and Destination point* instead of ***dragging*** the *Reference point*.

The status line allows you to enter the exact scale (**F**actor) in the status line.

Stretch Tool

This tool allows you to scale objects along one axis and a specified angle. You can stretch a copy when you press the **Ctrl** key (*Macintosh*: ***Option*** key). Pressing the **Shift** key you can select additional objects. If you select more than one object, they remain in the same position relative to each other

Stretching objects

1. Select the geometry you want to stretch.
2. Select the **Stretch** tool.
3. Click a point on the object that you want to remain stationary.
4. Click a second point on the object you want to move (*Reference point*).
5. Click a third point to indicate the new location of the *Reference point*.

The Status Line allows you to enter the exact *Scale factor* and *Angle*.

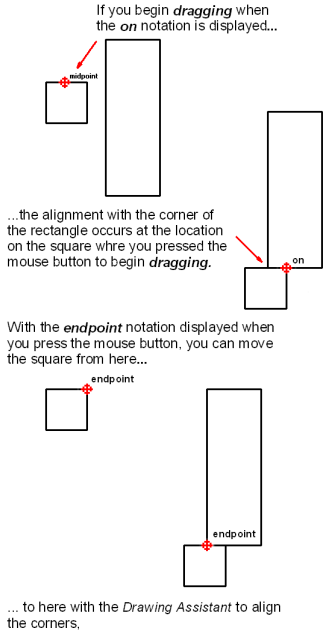
Moving Objects with Tools

When an object is selected, move the pointer near the object until the **Move** symbol appears and ***drag*** the object to a new location.

Using the Drawing Assistant for Moving

When the pointer becomes the **Move** symbol, you can ***drag*** the object around and see the *Drawing Assistant's* notations relative to the object's location when you pressed the mouse button.

In this way, if you move the pointer over a control point so the *Drawing Assistant* activates it, you can align the object you are moving with that point.

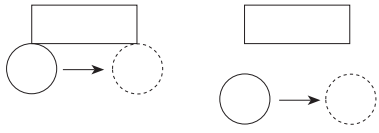

If you begin dragging when the on notation is displayed...
midpoint
...the alignment with the corner of the rectangle occurs at the location on the square whre you pressed the mouse button to begin dragging.
on
With the endpoint notation displayed when you press the mouse button, you can move the square from here...
endpoint
endpoint
... to here with the Drawing Assistant to align the corners,

Selected Move versus Move Tool

You can move objects either with the **Selection** tool or with the **Move** function.

Moving with the Selection Tool

The move function of the **Selection** tool allows free movement of the selected object.

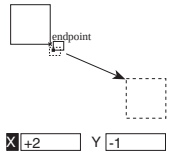


To move a circle from one location to another as shown above, use the **Selection** tool and the *Drawing Assistant* to align the 90° ***quadrant*** with the corner of the rectangle. The circles do not need to touch the rectangle to be aligned with it.

Moving with the Move Function

The **Move** tool (on the **Transformation** subpalette) allows you to move the selection with reference to other geometry.

Move a **1 cm** square **2 cm** in the x direction and **1 cm** in the y direction as in the following graphic:



1. Select the square to be moved.
2. Select the **Move** tool.
3. With the **Move** tool, click one corner of the square when the ***endpoint*** notation is displayed.
4. Enter in the **X** entry box on the status line **+2** and in the **Y** entry box **-1**.
5. Press the **Enter** key.
   The square is moved, similar to the graphic above.

A positive or negative value entered in the status line determines the direction along the **X** or **Y** axis. A negative value moves the object to the left or down on the screen. A positive value to the right or up.

Copying Objects with Tools

You can copy selections with the **Copy** command as discussed earlier in this section or by holding down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while using the following tools:

- Selection tool
- Single Line tool
- Center-Point and Opposite-Point Circle tools
- Ellipse tools
- Polygon tools
- Text tool
- Transformation tools

Copying with the Selection Tool

You can hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*) and ***drag*** a copy of the selection to a new location.

Copying with the Selection Tool

1. Click the **Selection** tool.
2. Select the object or objects you want copied.
3. Hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*).
4. ***Drag*** a copy of the selection to a new location.
5. Release the **Ctrl** key (*Macintosh*: ***Option*** *key*).

Copying with the Geometry Tools

You can make copies with the following drawing tools:

- Single Line tool
- Center-Point and Opposite-Point Circle tools
- Ellipse tools
- Polygon tools
- Text tool

Copying with the drawing tools

1. Construct the geometry you want to copy.
2. Hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*).
3. Click a new location. The click determines the location of the first point specified during the construction of the original geometry (the center of a center-point circle, for example).
4. Release the **Ctrl** key (*Macintosh*: ***Option*** *key*).

Copying with the Transformation Tools

You can hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*) while using a **Transformation** tool to make a copy of the selected geometry.

Copying with the Transformation Tools

1. Select the object you want to copy and transform.
2. Select one of the four **Transformation** tools.
3. Hold down the **Ctrl** key (*Macintosh*: ***Option*** *key*).
4. Perform the transformation according to the directions in the message line.
5. Release the **Ctrl** key (*Macintosh*: ***Option*** *key*).

   A copy is transformed and the original remains unchanged.

Copy Option versus Copy Command

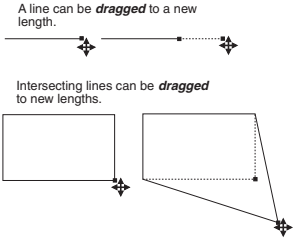
Usually, using the copy option of a geometry tool is faster than using the **Copy** and **Paste** commands in the **Edit** menu. The **Copy** command is very useful for copying to a different document or application.

Sizing Objects with Tools

Normally, you size an object with the **Selection** tool or the **Expand/Shrink** tool. In some cases it is useful to use the move tool for sizing objects.

Sizing an object with the Selection Tool

You can stretch objects by selecting a point and ***dragging*** it to a new location.



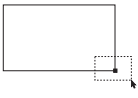
1. Be certain that **Selectable Points** is set in the **Edit** menu.
2. Click the **Selection** tool.
3. ***Drag*** a selection marquee around the *control points* that represent the area you want to stretch.
4. ***Drag*** the points to a new location.

Sizing an object with the Move Tool

You can also size an object with the **Move** tool. Using the **Move** tool allows you to specify the distance the selected point(s) should be moved by specific values along the **x** and **y** direction in the status line.

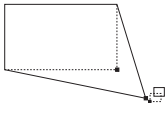
To move the corner of the rectangle in the next graphic with the **Move** tool you proceed as follows:

1. Be certain that **Selectable Points** is set in the **Edit** menu.
2. Click the **Selection** tool.
3. ***Drag*** a selection marquee around the lower right corner of the rectangle.



4. Click the **Move** tool.
5. Enter **+1** in the **X** entry box and **-1** in the **y** entry box in the status line.
6. Press the **Enter** key.

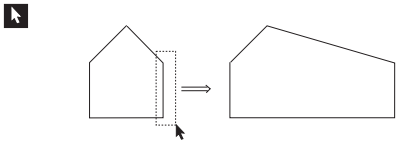
   The corner of the rectangle is moved the distance you have specified in the status line.



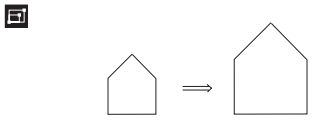
Use the **Move** tool for sizing objects only once since the next values you enter in the status line refer not to the new position of the moved point(s), but **always** to the original one. That means if you enter (in our example) **-1** in the **X** entry box and **+1** in the **Y** entry box, the point is not moved back to its original position, but in the opposite direction referring to its original position.

Selected Sizing versus Expand/Shrink Tool

Dragging a control point of a selected object changes the size of the object, but it also distorts the object, changing the proportion between height and width.



The **Expand/Shrink** tool on the **Transformation** subpalette enables you to resize geometry while maintaining its proportions.



Use the **Expand/Shrink** tool to enlarge or shrink a shape proportionally

In addition, you can specify proportions by clicking points on other geometry. For example, you can resize an object to fit within another object by clicking the boundary into which the resized object must fit.

Editing Commands

Four commands in the **Edit** menu allow you to change objects without changing the physical geometry. You can copy or move objects within the same documentation. You can also copy or move objects to other documents, even to other applications.

The difference between the **Cut** and **Copy** commands is that cutting a selection removes it from the drawing area while a copied selection remains in the drawing area.

The **Cut**, **Copy**, and **Paste** commands do not alter the object's attributes -the layer and pen style (*color*, *weight*, and *pattern*). If you want to change the attributes of an object, use the **Edit Objects** command in the **Edit** menu.

Cut Ctrl+X [Macintosh: ⌘X]

This command in the **Edit** menu removes the selected objects and places them on the *Clipboard*. Each selection you cut or copy to the *Clipboard* replaces the previous *Clipboard* contents.

Cutting objects

1. Select the object to be cut.
2. Choose **Cut** from the **Edit** menu.

Once you cut a selection, you can paste it. You can use **Cut** and **Paste** to move geometry or text around the document, from one sheet to another, or from one document to another. You can even paste the cut selection into a document in a different application.

You can also erase selections with the **Delete** or **Backspace** key or by choosing **Delete** from the **Edit** menu. Selections erased in this way do not go on the *Clipboard* and cannot be pasted. You can restore them, however, by using **Undo**.

Moving geometry with the Cut command

1. Select the objects you want to move.
2. Choose **Cut** from the **Edit** menu.
3. Display the location where you want the selection to appear in the drawing area, scrolling if necessary.
4. Choose **Paste** from the **Edit** menu.

The object appears in the center of the drawing area on the original layer on which it was created or onto the work layer if it's from a different application. The object is selected so you can move it, if you want.

If you want to move an object from one layer to another, use the **Edit Objects** command in the **Edit** menu.

Copy Ctrl+C [Macintosh: ⌘C]

This command in the **Edit** menu places a copy of the selection onto the *Clipboard* without deleting the original selection. You can paste the copy elsewhere in the current document or into a different document. You can even paste the copied selection into a document created with a different application.

You can also create a copy of an object or text box with the **Selection** tool. First select the object, then hold down the **Ctrl** key (*Macintosh*: **Option** key) and ***drag*** a copy of the object to a new location.

Copying objects

1. Select the objects to be copied.
2. Choose **Copy** from the **Edit** menu.

   The selection goes on the *Clipboard*, and it remains in the current document.

You can copy an object using any of the **Transformation** tools by holding down the **Ctrl** key (*Macintosh*: **Option** key) while using the tool.

Paste Ctrl+V [Macintosh: ⌘V]

This command in the **Edit** menu pastes a copy of the Clipboard contents onto the center of the drawing area. The Clipboard contents are not changed when you use the **Paste** command.

You can also paste the selection into another document or into a document created with another application.

If you want to create an even distribution of geometry, you can use **Linear Duplicate** or **Polar Duplicate** in the **Edit** menu.

If the object was cut or copied from a ***DraftBoard*** document, when it is pasted, it goes on the layer it was on when it was cut or copied. A pasted selection retains its original attributes. Objects cut or copied from non-***DraftBoard*** documents are pasted onto the work layer.

The Clipboard

The **Cut** and **Copy** commands both place a copy of the selected object on the *Clipboard*. The *Clipboard* is a buffer —a temporary storage place that holds the last cut or copied selection. The contents of the *Clipboard* are objects, not bitmaps.

Erasing Geometry

You can use several methods to erase objects:

- You can select the object(s), then press the **Backspace** key.
- You can retrieve what you deleted with the **Undo** command.
- You can undo the last eight actions.
- You can select the object(s), then choose the **Delete** command from the **Edit** menu. You can retrieve what you deleted with the **Undo** command.
- You can select the object(s), then choose **Cut** from the **Edit** menu. You can retrieve what you cut by using the **Paste** command, ***as long as you have not cut or copied anything else***.

Delete

This command in the **Edit** menu deletes a selection without placing it onto the *Clipboard*. You can **Undo** this deletion, but you cannot paste what is deleted. In addition to this command, you can use the **Delete** or **Backspace** key to delete a selection.

Retrieving with Undo and Redo

The number of **Undo's** and **Redo's** is limited to **32**.

Once an object is deleted or cut, you can retrieve it with the **Undo** command in the **Edit** menu. **Undo** can reverse each of the last eight actions. You can use **Ctrl+Z** (*Macintosh:* ***⌘Z***) to issue the **Undo** command from the keyboard.

Undo Ctrl+Z [Macintosh: ⌘Z]

This command in the **Edit** menu reverses the last action. You can **Undo** and **Redo** any of the last eight actions. For example, if you delete an object, you can choose **Undo** to restore it. You can then use **Redo** to return to the deleted version.

You can undo actions that create and edit geometry and text but not actions that do not change the contents of the drawing, such as resizing the window or quitting.

Redo Shift+Ctrl+Z [Macintosh: Shift+⌘Z]

If you are using a tool which involves a multistep process, such as constructing connected lines or creating a 3-point fillet, choosing **Undo** reverts to the beginning of the process. Some other activities can be stopped by pressing the **Escape** key.

This command in the **Edit** menu reinstates the last action reversed by **Undo**. You can **Undo** and **Redo** any of the last eight editing actions.

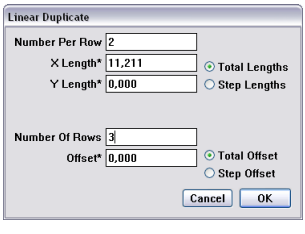
Duplicating Objects

DraftBoard allows you to create multiple copies of an object and arrange them in an array along a straight line or in a circular pattern.

Linear Duplicate

This command in the **Edit** menu allows you to create an array of copies along a straight line.

An asterisk (*) appearing beside an entry field indicates that you can fill in the box by ***dragging*** the mouse in the drawing area to indicate the length.



The **Linear Duplicate** dialog box contains the following settings:

Number Per Row The total number of objects you want in each row. Be sure to include the selected object itself in this total count.

X Length/Y Length	The **X Length** value determines the horizontal length of a row. The **Y Length** value determines the vertical rise of the row. The **X Length** and **Y Length** can be set by entering values, or by clicking the mouse in the drawing area.
Total Lengths	The **X** and **Y Length** is the distance from the selected object to the last copy.
Step Lengths	The **X** and **Y Length** is the distance from the selected object to the first copy.
Number of Rows	The total **Number of Rows** that you want.
Total Offset	The offset is the perpendicular distance between the row containing the selected object and the **last** row.
Step Offset	The offset is the perpendicular distance between the row containing the selected object and the **next** row.
Offset	The perpendicular distance setting for the **Total Offset** or **Step Offset**. You can specify the Offset by entering a value or by clicking the mouse in the drawing area.

Using Linear Duplicate

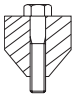
1. Select the object to be duplicated.
2. Choose **Linear Duplicate** from the **Edit** menu.
3. Enter the **number** of objects **per row**.
4. Specify **Total** or **Step** Lengths.
5. Click the **X Length*** box.
6. Move the pointer to the drawing area and drag to indicate the length of the row of copies.
   Both the **X** and **Y** values appear in the entry boxes.
7. If you want more than one row, enter the number of rows, and specify the type of offset.
8. Enter the offset, if any. You can use a negative number.
9. Click **OK**.

If you begin a duplication and discover that it is not what you want, you can stop the process by pressing the **Escape** key, and then choosing **Undo** to remove the copies from your document.

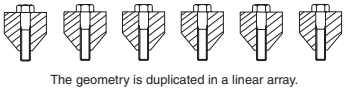
Dragging to indicate the offset is particularly useful if you want the offset to equal the distance between two objects.

An Example

Suppose you want to illustrate six machine screws. You could draw geometry, then use **Linear Duplicate** to copy the geometry in an array.



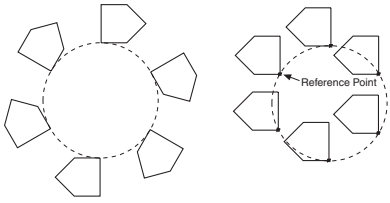
1. Create and select the geometry you want to duplicate.
2. Choose **Linear Duplicate** from the **Edit** menu.
3. Enter **6** in the **Number Per Row** box.
4. Click the **Step Length** button.
5. Click in the **X Length** box.
6. Move the pointer into the drawing area and drag across the widest point of the step.
7. Click at the end of the entry in the **X Length** box.
8. Type ***2** to multiply the width of the geometry by two.
9. Click **OK**.



The geometry is duplicated in a linear array.

Polar Duplicate

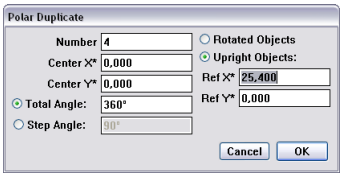
This command in the **Edit** menu allows you to copy and rotate the selected geometry. When you duplicate an object in a circular array, you can specify the number of duplications, the center of the array and whether the objects are copied in a rotated orientation or an upright orientation.



Rotated Objects Upright Objects

When you select the **Polar Duplicate** command in the **Edit** menu the following dialog box is displayed:

An asterisk (*) appearing beside an entry field indicates that you can fill in the box by ***dragging*** the mouse in the drawing area to indicate the length.



The **Polar Duplicate** dialog box contains the following settings:

Number — The total number of objects you want. Be sure to include the selected object itself in this total count.

The **X/ Y** values can be also dragged on the drawing area

Center X/ Center Y — The **Center X** and **Y** values determine the center of the circular array.

If the copies should be arranged semicircular the *Total Angle* must be **180°**.

Total Angle — The total number of degrees between the center of the selected object and the center of the last copy.

If you want to place an object three times in a semicircle, the *Number* must be **3** and the *Step Angle* **90°**.

Step Angle — The number of degrees between the center of the selected object and the center of the first copy.

If the option **Rotated Objects** is selected, in principle any point on the object becomes a *Reference Point*.

Rotated Objects — Copies are rotated relative to the angle between the copies. Each single arbitrary point of all copies have the same distance to the center of the circular array. That means that any point of the object can be the *reference* point.

Upright Objects — Copies are upright with respect to the selected object. This option requires a reference point.

The **Ref X**- and **Ref Y** coordinates can be specified by entering the values in the respective enter fields or clicking a point on the drawing area.

Ref X/Ref Y — The *reference* point for the **Upright Objects** option.

Ref X and **Ref Y** determine an imaginary point duplicated around the center, as specified. The selected objects are reproduced in the same position relative to each imaginary point that is duplicated. With the option **Rotated Objects** the reference point has a constant distance to the center of the circular array.

Using Polar Duplicate

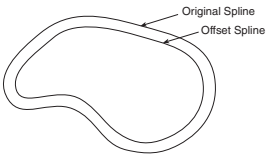
If you begin a duplication and discover that it is not what you want, you can stop the process by pressing the **Escape** key, and then choosing **Undo** to remove the copies from your document.

1. Select the object to be duplicated.
2. Choose **Polar Duplicate** from the **Edit** menu.
3. Enter the **number of objects** in the circular array.
4. Click the **Center X*** box.
5. Move the pointer to the drawing area and click to indicate the center for the array of copies.

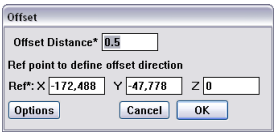
   The values for **Center X** and **Center Y** appear in the entry boxes.
6. Specify **Upright** or **Rotated** objects by clicking a button.
7. If you specified **Upright**, click a location for the reference point in the drawing area.
8. If you do not want the copies in a complete circle, click the **Total** or **Step Angle** button.
9. Enter a value for the specified angle type.
10. Click **OK**.

Offset

This command in the **Edit** menu allows you to create offsets from *lines*, *arcs*, *circles*, *ellipses* and *splines*.



When you select the **Offset** command in the **Edit** menu the following dialog box is displayed:



The asterisk indicates that you may use the cursor to input an offset distance by ***dragging*** between any two points on the drawing area. The cursor must be in the **Offset Distance** field while ***dragging***.

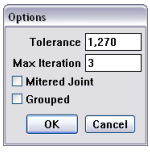
The **Offset** dialog box contains the following settings:

Offset Distance The offset distance is the distance away from the original geometry that the new geometry will be created. For multiple offsets additional values may be entered manually into the **Offset Distance** field, separated by semicolons.

The offset **Direction** is also filled in the respective entry fields when you ***drag*** the offset distance since the vector you ***drag*** determines the **Offset Direction** automatically.

Ref. The **X** and **Y** values can be filled in manually or indicated by a mouse click on the drawing area offset from the object (the **Z** value must be **0**). The point clicked determines which side of the object the offset should be performed.

Clicking the **Options button** displays the following dialog box:



The **Options** dialog box contains the following settings:

Tolerance For *Spline* and *Ellipse* objects the offset distance is approximated by a spline. By setting a **Tolerance** for this type of object you can determine how accurate the offset is relative to its theoretical position.

Max Iteration ***DraftBoard*** will iterate when calculating the offset of *Spline* and *Ellipse* objects to improve the approximation the number of times indicated in the **Max Iteration** field. The default value is **3**. Setting a high value will decrease performance.

Mitered Joint If you mark this option the offset object will have square external corners instead of filleted corners. This option is by default not selected. The radius of the filleted corners is automatically set to the offset distance. For instance; a box that is offset by **.25** will have a fillet at each corner with a radius of **.25**.

Grouped If this option is marked the new offset geometry will be grouped when created. This option is by default not selected to produce ungrouped geometry.

The geometry will be only grouped, if the **Grouped** option was checked.

Creating an Offset

1. Select the object you want to offset.
2. Select the **Offset** command in the **Edit** menu.
3. Click into the **Offset Distance** field.
4. Input an offset distance by ***dragging*** between any two points on the drawing area.
5. Click into one of the **Ref** entry fields.
6. Input an offset direction by ***dragging*** between any two points on the drawing area.
7. Click the **Options** button.
   Specify in the **Options** dialog box a **Tolerance**, if needed and whether or not the object will be offset with mitered corners. The new geometry will be created as a group and will leave the original geometry selected.

Changing the Characteristics of Objects

You can change a single characteristic faster by simply selecting the object and changing the characteristic. But if you are changing several characteristics or any measurements or positions, you should use the **Edit Objects** command.

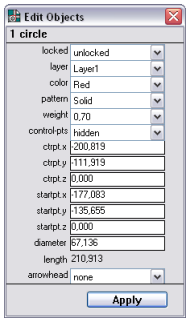
You can edit selected objects by either changing individual characteristics, such as layer or line style, or by changing the specifications in the **Edit Objects** dialog box. The **Edit Objects** dialog box also provides information about the selected object in addition to allowing you to make changes.

Edit Objects Ctrl+I (Macintosh: ⌘I)

This command in the **Edit** menu allows you to edit selected objects by changing individual characteristics, such as *length*, *layer*, or *pen style*, or other specifications. Changes made through this dialog box can be reversed with the **Undo** and **Redo** commands.

Editing objects

1. Select the object to be edited.
2. Choose **Edit Objects** from the **Edit** menu.
   The dialog box appears.



Many boxes have pop-up menus for selection. Press the down arrow and then ***drag*** to the selection you want.

3. Change the information in the entry boxes.
   Double-click the entry box and type a new entry (do **not** press **Enter**).

In the **Edit Objects** dialog box you can use mathematical operators. A list of all allowed operators is in the *Appendix* of this manual.

4. Make any other changes you want.
5. Click the **Apply** button.
6. Double click the **Close** box in the upper left corner to close the **Edit Objects** dialog box.

You can use **Undo** and **Redo** to reverse changes made through this dialog box.

The specifications shown in this box depend on the type of object selected, and include at least the following:

- Number (or type) of objects
- Lock status
- Current layer
- Current color
- Current pattern
- Current weight
- Absolute coordinates for the starting point and ending point of the object

You can use different units in each entry box like inch (''), feet ('), feet/inches (x'y''), centimeter (**cm**) millimeter (**mm**) or Meter (**m**). You are also allowed to mix English and metric units as long as they are labelled correctly like 10'' + 25.4 cm.

The measurements reflect the settings of the **Units** option in the **Preferences** submenu. If you want to prevent changes to an object, you can specify **locked** in this dialog box, or you can select the object and choose **Lock** from the **Arrange** menu.

If you are doing a lot of editing, you may want to leave the **Edit Objects** dialog box displayed. In that way you can select the object, make the changes in the dialog box, click **Apply**, and then go on to the next object.

If more than one object is selected when you choose **Edit Objects**, only the common information is displayed. Entries are blank when the information isn't common. For example, if you select two concentric circles, the center will be displayed in the **Edit Objects** box but the diameter box will be blank.

If you select several objects and then choose **Edit Objects**, all of the objects will reflect the changes you make in the dialog box. For example, if you select all dimensions and change the text entry in the Edit Objects dialog box to **2**, all the dimensions will display a **2**.

Moving an object to a different layer

1. Select the object(s) to be moved.
2. Choose **Edit Objects** from the **Edit** menu.
3. Click the arrow for the **layer** entry box.
4. Click the layer from the list.
5. Click **Apply**.

Arranging Geometry

The **Arrange** menu contains some commands that enable you to subdivide geometry, group geometry so it behaves like a unit and lock geometry to prevent changes . In addition, this menu contains a command to refresh the screen after you have changed geometry.

This menu contains also commands to sort objects on the screen. When using these commands objects in the foreground will cover objects in the background. This feature allows to manipulate the default drawing order of ***DraftBoard***, that puts always that object into the foreground that was drawn last.

The display order set with the **Bring to Front** and **Send to Back** commands is valid as well when selecting objects.

Bring to Front

This command brings a selected object into the foreground and will cover all underlying objects. When this command is selected for another object, the object brought into the foreground before, will go one position backwards. This allows to sort objects specifically on the screen. If several sorted objects are grouped, the sorting order will be kept, even if the grouped object is send to front again.

This command can be used as well when placing text in front of a filled surface.

Send to Back

This command sends a selected object into the background and will be covered by all objects lying in the foreground. When this command is selected for another object, the object brought into the background before, will go one position forwards.

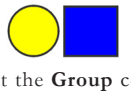
This allows to sort objects specifically on the screen. If several sorted objects are grouped, the sorting order will be kept, even if the grouped object is send to back again.

With multiple hierarchical grouped objects that functions only partly.

Bringing objects to Front/Back (Example)

1. Draw a circle using the line color **Black** and a line weight of **1,5 mm**.
2. Fill the circle with the color **Yellow**.
3. Select the yellow fill area (without the black perimeter).
4. Select the **Send to Back** command in the **Arrange** menu.
   The yellow fill area is send to back and covered by the black perimeter circle.

5. Select the circle and the fill area and choose the **Group** command in the **Arrange** menu.
   The yellow fill area and the circle perimeter are grouped as sorted.
6. Draw a rectangle with the color **Black** and a line weight of **1,5 mm**.
7. Fill the rectangle with the color **Blue**.
8. Select the blue fill area (without the black rectangle line).
9. Select the **Send to Back** command in the **Arrange** menu.
   The blue fill area is send to back and covered by the black perimeter lines.



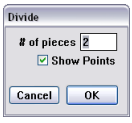
10. Select the **Group** command in the **Arrange** menu.
    The blue fill area and the perimeter lines are grouped as sorted.
11. Select the blue rectangle and move it partly over the yellow circle.
12. Deselect the blue rectangle by clicking anywhere onto the drawing area.
    The blue rectangle in the foreground covers the yellow circle.

13. Select the yellow circle.
14. Select the **Bring to Front** command in the **Arrange** menu.
    The yellow circle is brought to front and covers the blue rectangle.

Divide

This command in the **Arrange** menu subdivides the selected geometry into the specified number of equal parts.

1. Select the object to be divided.
2. Choose **Divide** from the **Arrange** menu.
   The following dialog box appears.



3. Enter the number of equal parts you want to divide the selection into.
4. Click **OK**.

You can see the divisions when you display the points by clicking **Show Points** in this dialog box or choosing **Show Points** from the **Layout** menu.

Group Ctrl+Y [Macintosh: ⌘Y]

This command in the **Arrange** menu combines selected objects to function as a single object.

Grouping objects

If you move the group, all components move together. If you change the size of a group, the individual objects change proportionally.

1. Select the objects to be grouped.
2. Select **Group** from the **Arrange** menu.

Once you group geometry, you can't edit the individual objects within the group unless you ungroup them.

You can also combine groups. For hierarchical groupings, ***Draft-Board*** ungroups each group in the order in which they were combined.

Changing the members of a group without adding a hierarchical level

To create temporary groups, you can ***drag*** a selection marquee around several objects to treat them as a single unit while they are selected.

When you want to add new geometry to an existing group, and you select the group and the new geometry and choose the Group command, you end up with a group within a group. If you want to make a single group, follow these directions:

1. Select the group.
2. Choose **Ungroup** from the **Arrange** menu.
   The geometry is ungrouped with the individual objects selected.
3. Hold down the **Shift** key and select the geometry you want to add to the group.
4. Choose **Group** from the **Arrange** menu.

You can follow a similar procedure to remove members of a group.

Ungroup

This command in the **Arrange** menu separates selected grouped objects into their individual components.

Ungrouping Objects

1. Select the group.
2. Choose **Ungroup** from the **Arrange** menu.
   The individual objects in a group are selected.

Lock

This command in the **Arrange** menu prohibits editing or movement of selected geometry.

Preventing accidental changes to one or more objects

1. Select the objects to be locked.
2. Choose **Lock** from the **Arrange** menu.

Although you cannot move or change a locked object, you can copy, group, and select it.

Preventing changes to an entire document

1. Choose **Select All** from the **Edit** menu.
2. Choose **Lock** from the **Arrange** menu.

Selected objects can also be locked or unlocked by clicking the **Locked** box in the **Edit Objects** dialog box.

Unlock

This command in the **Arrange** menu removes the lock in the selection. You can change and move the unlocked objects.

Unlocking Objects

1. Select the locked object(s).
2. Choose **Unlock** from the **Arrange** menu.

Selected objects can also be locked and unlocked by clicking the **Locked** box in the **Edit Objects** dialog box.

CHAPTER

7

Text, Fills and Crosshatches

Text

AutoText

Crosshatching and Solid Fills

Text, Fills and Hatches

This chapter describes the features you can use to add detail to your drawing: *text*, *crosshatching* and *solid fills*.

Text

When you want to annotate your drawing, you'll use the **Text** tool to create a text box for entering characters from the keyboard. You can create, import, and edit text on screen. You can use the **Text** menu to set the characteristics, such as *font*, *text size*, *style*, *alignment*, *line spacing*, and *indentation*.

Text Tool

This tool on the tool palette allows you to create and edit text on screen. The text you enter has the characteristics set in the **Text** menu.

Creating text

1. Select the **Text** tool from the tool palette.
2. Create a text box by ***dragging*** or by clicking two locations to place the opposite corners of the text area.

   The text box which appears is as tall as a single line of text and as wide as you indicate. The text cursor appears in the box, ready for your typing.
3. Enter the desired text from the keyboard.

   Pressing the **Delete** key erases characters to the left of the cursor.

   The text you enter automatically *word wraps* to the next line when it reaches the right side of the text box. If you resize the text box, the words rewrap automatically.

Importing a text file

You can import an **ASCII** file that contains text you want to use in your drawing. For example, you can place specifications you've written with a word processor on your drawing without retyping it. The entire file is imported, so if you want to use only part of the document, create a new file that contains only the information you want entered on the drawing.

An **ASCII** text file is usually created by word processing software and saved as *plain text*. It usually contains only standard keyboard characters (*letters*, *numbers*, *punctuation*, and *spaces*) without the formatting typically found in word processing documents.

1. Select the **Text** tool from the tool palette.
2. Create a text box by ***dragging*** or clicking twice to place the opposite corners of the text area.
3. Choose **Import** from the **File** menu.

   A dialog box appears.
4. Select the text file you want to import.

   The **Import** dialog box appears and the **Text** file type is specified.
5. Click **OK**.

   The text file appears in the text box you created.

Editing text

If you want to change characteristics for the entire text entry, use the **Selection** tool to select the text block.

1. Select the **Text** tool from the tool palette.
2. Move the pointer directly over the text to be edited.

   The pointer turns into an **I-beam** text cursor when it is over text. (This only occurs when the **Text** tool is active.)
3. Use the text cursor to select text.

   You can ***drag*** either direction to select characters, and double-click on an individual word to select the word.

   You may also simply click to place the text cursor within the existing text if you want to add more to it.
4. Modify the selected text by choosing the appropriate commands from the **Text** menu.

Changing the characteristics of a text block

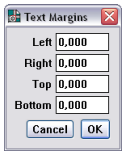
If you want to change the characteristics of a word or portion of a text block, use the **Text** tool to select what you want to change.

You can change the *font*, *text size*, *style*, *spacing*, and *indentation* of a text block.

1. Use the **Selection** tool to select the text block to be changed.
2. Select the new characteristic from the **Text** menu.
   The selection shows the new characteristic.

Modifying text margins

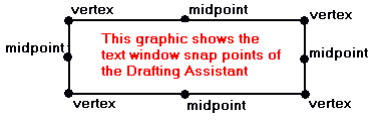
1. Choose the **Text Margins** command from the **Preferences** submenu in the **Layout** menu.
   The following dialog box is displayed.



2. Enter values for the *Left*, *Right*, *Top* and *Bottom* text margins.
   This values are valid for all future text placed with the **Text** tool from the *Function palette*. You can save these margins permanently using the **Save Preferences** command from the **Preferences** submenu in the **Layout** menu.

Moving Text

Each text object has **nine** snap points for the *Drawing Assistant*. These snap points will help you to move and/or align text with the **Selection** tool.



1. Choose the **Selection** tool from the Tool palette.

2. Select the text object you want to move.
3. Move the cursor over one of the nine text snap points. The cursor becomes a ***4-way Move*** *symbol.*

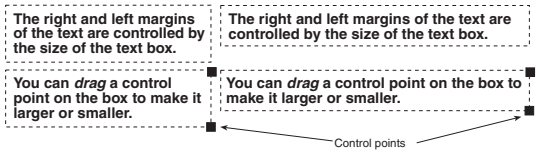
4. Place the ***4-way Move*** *symbol* at the desired snap point using the *Drawing Assistant* notations as a guide.
5. ***Drag*** the text object to the new location.

Changing the size of the text box

Selectable Points in the **Edit** menu must be active for selecting the control points of the text box.

When you want to set the size of the text box accurately you select the text box with the **Selection** tool and choose the **Edit objects** command in the **File** menu. In the **Width** and **Height** entry boxes you can enter values for the box size.

1. Click the **Selection** tool.
2. ***Drag*** a selection marquee around the right side of the text box.
3. ***Drag*** the control points to the left or right.
   The area changes size and the text rewraps automatically.


The right and left margins of the text are controlled by the size of the text box.
The right and left margins of the text are controlled by the size of the text box.
You can drag a control point on the box to make it larger or smaller.
You can drag a control point on the box to make it larger or smaller.
Control points

Moving Text and Files into the DraftBoard drawing area (Drag&Drop)

Text can be moved directly from all programs that support **OLE** into the ***DraftBoard*** *drawing area.*

DraftBoard *Files* (documents with the file extension **.vlm**) will be directly opened, when moved from the ***Windows****-Explorer* or from the ***Windows*** *Desktop* into the ***DraftBoard*** *drawing area.*

Dragging text into the DraftBoard Drawing area

1. Create text using a text editor (such as ***MS****-Word)* that supports **OLE**.

2. Select the text and ***drag*** it with the left mouse button pressed into the ***DraftBoard*** *drawing area*.
3. Release the left mouse button.

   The text is placed on the ***DraftBoard*** *drawing area* at the location where the mouse button was released. The text is displayed using the current ***DraftBoard*** text font and all paragraph formats will be maintained.

Dragging DraftBoard Files into the DraftBoard Drawing area

1. Select a ***DraftBoard*** *File* in the ***Windows*** *Explorer* or on the ***Windows*** *Desktop*.
2. ***Drag*** the ***DraftBoard*** *File* with the left mouse button pressed into the ***DraftBoard*** *drawing area*.
3. Release the left mouse button.

   The ***DraftBoard*** *drawing* is automatically opened in ***DraftBoard*** .

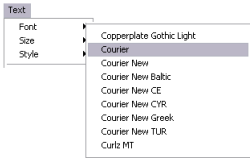
Text Menu

The **Text** menu contains commands for changing the *text font*, *size*, and *style*, as well as the *justification*, *spacing*, and *indentation*.

A *checkmark* in front of a menu item shows the current attribute.

Font

This command in the **Text** menu changes the *font* for selected text or future text entries.



Text can be rotated with the **Rotate** function like any other object in a drawing including text using *Postscript* and *True Type* fonts.

The fonts listed in the menu are those installed on your computer and include ***DraftBoard's*** plotter fonts.

Use ***Plotter*** *fonts* whenever you are creating text for a drawing that you intend to send to a plotter, since plotting *Postscript* or *TrueType* fonts take much more time to print or may be substituted.

The font you choose stays in effect for the current document until you choose another font.

Special characters and accents are available as described in the *Appendix* of this manual.

Extended font selection

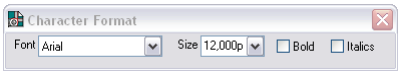
The **Font** submenu can display up to *20 fonts*. If you have installed more than 20 fonts select **More** in the **Font** submenu of the **Text** menu to get access to all available fonts.

Choosing **More** from the **Font** submenu brings up a dialog box allowing you to specify the *font*, *text size* and *text style*. The text size is measured as point size. If you want to specify the size in the units you have set for your drawing, you have to type the exact unit such as **"** for inch or **mm** for millimeter after the value.

In addition to the plotter fonts shipping with ***DraftBoard*** you can use all plotter fonts that are compatible with the file format **.shx** of AutoCAD©. When you copy these files into the **Fonts** folder of the ***DraftBoard*** folder, the new fonts are displayed in the **Fonts** submenu after relaunching ***DraftBoard***.

1. Choose **More** from the **Font** submenu in the **Text** menu.

   The **Character Format** dialog box displays:



2. Select the character format. The text size is always measured as point size (**pt**). If you want to specify the size in another unit like in **Millimeter**, include **mm** with your numeric entry.

All selections made in this dialog box take effect immediately for all selected text or text boxes.

All menu items in the **Text** menu affect only text created with the **Text** tool. Text attributes for dimensions are set in the *dimension editor* under **Preferences** in the **Dimension** menu.

However when you type in a value (only possible in the **Size** box) the change becomes effective only until you do another selection or continue to work on the drawing area.

You can leave this dialog box open to assign other character formats. If you want to close this dialog box, you have to double-click the **Close box** in the upper left corner of the dialog box.

The character format you specify stays in effect until you choose another character format.

Printing TrueType Fonts on Pen Plotters

When you want to print ***TrueType*** *Fonts* on pen plotters you must use a ***WinLINE Plus*** *Plotter Driver*. Look for more information about ***WinLINE*** *Plus* in the Internet under ***www.winline.com***.

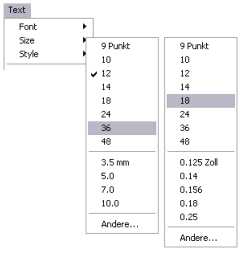
Font Sizes and ANSI Standards

A typical font specified for **7,0** mm may produce a character that is only **3,5** mm. The variation depends on the proportional spacing of the font. If you want a font set to exact specifications, use a **Plotter** font, which conforms to **ANSI** standards.

Size

This command in the **Text** menu sets the font size for the selected text or for future text entries in the current document. The size can be specified as either points or units, as defined in the **Units** dialog box of the **Preferences** submenu in the **Layout** menu.

If you have selected *American* units under **Units** in the **Preferences** submenu of the **Layout** menu the *text size* is displayed in **Point** and **Inches**, when you have selected *metric* units it is displayed in **Point** and **Millimeters**.



Specifying a non-standard text size

Choosing **Other** from the **Size** submenu brings up a dialog box allowing you to specify the exact text size you desire in whatever units you've set for your drawing. You can specify point size by entering **pt** after the value in the entry box; for example, if you enter **11 pt** the font size will be 11 point.

All items in this dialog box affect only text created with the **Text** tool. Text attributes for dimensions are set in the *Dimension Editor* under **Preferences** in the **Dimension** menu.

1. Choose **Other** from the **Size** submenu in the **Text** menu.

2. Enter the size you want. The size is measured in the current units as set in the **Preferences** submenu. If you want to specify a point size, include **pt** with your numeric entry.
3. Click **OK**.

   The size you specify stays in effect until you choose another size.

Text Size and Scaling

It is advisable to use the **Selection mask** to select all text and dimensions in case you have to assign the accurate text size **after** scaling.

A detailed description about *text size* and *scaling* can be found in ***Chapter 9*** *Basic Drafting*.

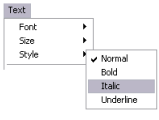
When you scale a drawing using the **Drawing Size** command without the options *keep text size* and *keep dimension text size* after you have added *text* and *dimensions*, you have to assign the correct text size again in the **Text** and **Dimension** menus.

This is not necessary when you add *text* and *dimensions* **after** scaling or scale the drawing with the options *keep text size* and *keep dimension text size* active.

Style

Certain **Styles** are only available on the *Macintosh*, not under *Windows*.

This command in the **Text** menu sets the style (such as *bold* or *underline*) for selected text or future text entries.



The style you specify stays in effect until you choose another style. A check (✓) indicates the current style. You can change the default setting by saving the changes in the preferences file ***Prefs.vlm*** (*Macintosh:* ***DraftBoard Prefs***) with the **Save Preferences** command in the **Preference** submenu of the **Layout** menu.

Text Alignment

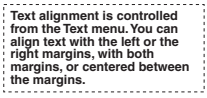
The four commands in the second section of the **Text** menu align your text within the text box. The text can be aligned on the *left*, the *right*, *centered* in the *middle*, or it can be fully *justified* so that it is aligned with both left and right margins of the text box.

If the **Text** tool is active, both the selected text and future text are aligned as specified, but existing text is not aligned. If you choose this command when a tool other than the **Text** tool is active, only future text entries are aligned. If a text entry is selected when you choose this command, the selection and future text entries are aligned as specified.

Align Left

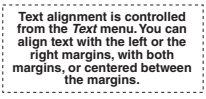
This command in the **Text** menu aligns the selected and future text at the left margin of the text area.

When you want to set the *text offset* from the box boundary you have to select the text box with the **Selection** tool and choose the **Edit objects** command in the **File** menu. You can enter values for the *text offset* in the related boxes.


Text alignment is controlled from the Text menu. You can align text with the left or the right margins, with both margins, or centered between the margins.

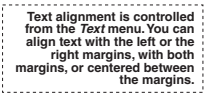
Align Middle

This command in the **Text** menu centers the selected and future text in the text area.


Text alignment is controlled from the Text menu. You can align text with the left or the right margins, with both margins, or centered between the margins.

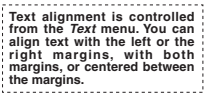
Align Right

This command in the **Text** menu aligns the selected and future text at the right margin of the text area.


Text alignment is controlled from the Text menu. You can align text with the left or the right margins, with both margins, or centered between the margins.

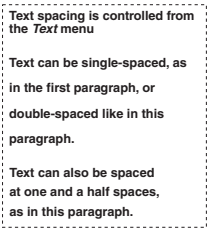
Justified

This command in the **Text** menu aligns both the left and right margins of the selected and future text in the text area.


Text alignment is controlled from the Text menu. You can align text with the left or the right margins, with both margins, or centered between the margins.

Text spacing

The **Text** menu has commands for *single spacing*, *double spacing*, or using *a space and a half* between the lines of your text.


Text spacing is controlled from the Text menu
Text can be single-spaced, as
in the first paragraph, or
double-spaced like in this
paragraph.
Text can also be spaced
at one and a half spaces,
as in this paragraph.

You can change the default setting by saving changes in the **Preferences** file.

The check (✓) in the **Text** menu indicates the current line spacing.

Single Space

This command in the **Text** menu sets the spacing of selected and future lines of text so each line occupies one space in the text area.

1-1/2 Space

This command in the **Text** menu changes the spacing of selected and future text so each line occupies one and a half spaces in the text area.

Double Space

This command in the **Text** menu sets the spacing of selected and future text so each line occupies two spaces in the text area.

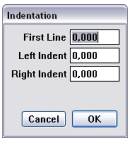
Indentation

If you wish to indent the text on the right or left side of the text box, or if you wish to have the first line indented, choose **Indentation** from the **Text** menu.

Indentation

This command in the **Text** menu sets the indentation of paragraphs for the selected text area as defined by the units set in the **Preferences** submenu.

1. Choose **Indentation** from the **Text** menu.
   The dialog box appears.



2. Specify the indentation you want. You can use point values if you include **pt** in your entry.
3. Click **OK**.

You can specify the number of units for any or all the choices:

First Line Sets the number of units for the indentation of the first line of each paragraph.

This is an example of a paragraph with the first line indented.

Left Indent Sets the number of units for the indentation of the left margin of each paragraph.

2. Dieser Absatz wurde links eingezogen.

Right Indent Sets the number of units for the indentation of the right margin of each paragraph.

This is an example of a paragraph with a right indentation.

AutoText

The two ***AutoText*** *commands* **TextBlocks** and **Forms** in the **Text** menu simplify and accelerate the placing of text blocks.

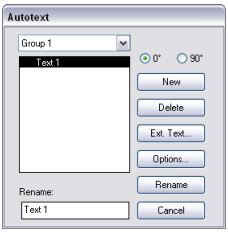
They allow you to define often used annotations as *Standard Text* that can be placed with only one mouse click in a drawing and to fill in and edit *Title Blocks* in a dialog box. In addition you can create your own *Standard Drawing formats* and *Title Blocks* or modify existing ones.

Defining Text blocks

Using the **TextBlocks** command you can define often used annotations as standard text blocks, group annotations in logical groups and place predefined annotations directly on the drawing area.

TextBlocks

The **TextBlocks** command displays the following dialog box.



The **TextBlocks** dialog box contains the following settings:

New When you click the **New** button a new *annotation* or *text group* is created depending which list item is active.

When a annotation is selected while clicking **New** a new annotation is created for the selected group with the name **Text 1**.

Clicking **New** when a *text group* is selected a new *text group* with the name **Group 1** and a new annotation for the new group named **Text 1** is created.

You can overwrite the predefined names **Text 1** or **Group 1** with a more suitable name in the **Rename** entry field.

Delete When you click the **Delete** button the selected annotation or text group is deleted. When you delete a text group all related annotations are deleted as well.

Rename With the **Rename** button you can rename the selected *annotation* or *text group*.

Options With the **Option** button you display a dialog box listing selectable attributes such as a *border frame* or *position line* for the selected text block and define the text style.

Attributes you assign to selected text are not saved with the selected text and are only valid for the current selection.

Extensions In the **Extended** dialog box you can assign an imported text to a selected text block.

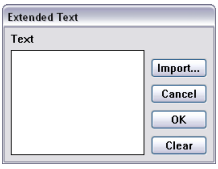
Text that contains imported text has a checkmark in front of the text name.

90° When this option is marked the annotation is placed *vertically* on the drawing area.

0° When this option is marked the annotation is placed *horizontally* on the drawing area.

Extended dialog box

When you select an annotation in the **TextBlocks** dialog box and click the **Extended** button the following dialog box is displayed:



In this dialog box you can assign to the selected annotation an extended text that is placed on the drawing instead of the selected text block. You cannot type *extended text* but only imported it as *ASCII-Text* with the **Import** button.

ASCII Text can be created with any text editor. You only have to specify ***ASCII*** *format* when saving the text. Take care that there is no empty text line at the end of the ASCII text since it will be imported as well.

Annotations that contain extended text have a checkmark in front of the name.

The width of the placed text frame equals the longest text line of the imported ***ASCII*** *file*. Text attributes such as *Text style, Text size* for all text blocks are set in the **Options** dialog box.

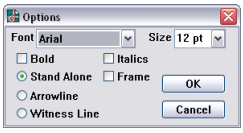
Editing extended text

Imported text cannot be edited in the **Extended Text** dialog box instead the original ***ASCII*** *file* has to be modified with a text editor. For an optimized organization it is recommended to place all imported texts in the **Text** folder of the **DraftBoard** folder in where you find some pre-installed extended text that however is not referenced to any notation.

Only the directory path of the imported text file is saved with the ***DraftBoard*** *file*, not the extended text itself. In case you cannot open the extended text example shipping with ***DraftBoard***, delete it in the **Extended Text** dialog box and import it again from the **Text** folder in the **Draft-Board** folder.

Options dialog box

When you click the **Options** button while a *Text block* is selected the following dialog box is displayed:



In this dialog box you can specify whether you want to place the annotation in a *border frame*, with a *position leader* or an *arrow line*.

For all marked annotations you can specify the *Text fonts*, the *Text size* and the *Text style* such as *bold* or *italic*.

Creating and placing Text blocks

1. Choose the **TextBlocks** command in the **Text** menu.
2. Create a new *Text Group* with a new *Text Block* by selecting first a *text group* and clicking then the **New** button. You can overwrite the automatically created names **Group 1** and **Text 1** using the **Rename** button.
3. In order to assign to the newly created *text block* an extended text you must click the **Extensions** button.

   The **Extensions** dialog box is displayed.

ASCII Text you import as extended text has not to be in the **Text** subdirectory of the **DraftBoard** folder but it is recommend for later editing of the extended text.

4. Import an ***ASCII*** *text* you have created before with an text editor from the **Text** folder of the **DraftBoard** folder into the **Extended** dialog box using the **Import** button.
5. Click **OK**.

   The **Extended** dialog box is closed. A check mark indicates that the text block contains an extended text
6. Click the **Options** button.
7. Specify the desired *options* and *text attributes* such as *text font* etc.
8. Click **OK**.

   The **Options** dialog box is closed.

The *Arrow type* for the position line you choose with the **Arrow** command in the **Pen** menu.

9. When you place a text block with a border frame you have to ***drag*** a vector on the drawing area. The *Starting point* of the vector defines the starting point of the *position line* and the *End-point* of the vector defines the position for the nearest corner of the text border and the endpoint of the position line.

   When you have marked the option **Stand alone** you click a point on the drawing area for the lower left corner of the text.

Die *Width* of the border frame equals the longest text line of the imported *ASCII Text*.

   The text block is placed *horizontally* or *vertically* on the drawing area as soon you release the mouse button. The orientation is depending on the marked option (**0°** or **90°**)
10. Place some more text blocks or close the **Text Blocks** dialog box clicking the **Close** box at the upper corner of the dialog box.

    When you select any function or command while the **Text Blocks** dialog box is open the dialog box closes automatically.

Editing Text Blocks in the drawing

Placed *text block*s can be only edited using the **Text** function in the ***DraftBoard*** function palette, not with the **TextBlocks** command.

Extended texts saved in the **Text** folder of the **DraftBoard** folder can be only edited with a *Text Editor*.

Title Blocks

When you import a drawing format you have to mark the option **unscaled** in order to place the drawing format at it's original size onto the drawing area. If you cannot see the imported drawing format you have to select the **Zoom all** command in the **Arrange** menu. If you have placed by mistake two drawing formats in a drawing all entry fields of both title blocks are displayed in the **Title Block** dialog box.

With the **Forms** command in the **Text** menu you can fill in the title blocks of all standard drawing formats that are saved in the **Format** folder of the **DraftBoard** folder.

As long as no *standard drawing format* is placed in a drawing the **Forms** command is greyed. It can be only selected if you place a drawing format with a *Title block* using the **Import** command in the **File** menu or the **Sheet into View** command in the **Views** menu.

DraftBoard allows you to create your own drawing formats that you can use with the **Forms** command. This procedure is described in *Chapter **10** Structuring Drawings*.

Forms

When you select this command in the **Text** menu a dialog box is displayed that contains all entry fields of the selected drawing format.

Filling in a Title Block

1. Place a *Standard drawing formate* from the **Layout** folder of the **DraftBoard** folder using the **Import** or **Sheet Into View** command.
2. Choose the **Forms** command from the **Text** menu.
   The displayed **Forms** dialog lists all entry fields of the title block.
3. Type in the desired parameters into the related entry fields. You don't have to fill in all entry fields.
4. Click **OK**.
   The **Forms** dialog box is closed and all entry fields of the selected title block are filled in according to your specifications.

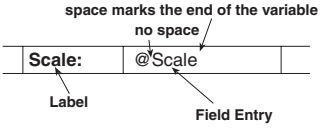
Editing Title Blocks

1. Select the **Forms** command from the **Text** menu.
   The **Forms** dialog box is displayed.
2. Modify the entries of the title block.
3. Click **OK**.
   The **TitleBlocks** dialog box is closed and the selected title block modified accordingly.

Creating Title Blocks

Before you can use the **Forms** command in the **Text** menu to fill in your own drawing formats you have to prepare the title blocks of these formats.

To enable ***DraftBoard*** to identify the title blocks all *field label names* have to be entered into the entry fields with an **@** character in front. For a possible entry field **Scale** you have first to put the label *Scale* in front of the entry field and then type in **Scale** into the entry field with a **@** character in front as shown in the following graphic:



In addition all label entries have to be placed on the layer **TitleBlocks**.

When you call the **Forms** command ***DraftBoard*** will check if there are any text entries on the layer **TitleBlocks** with an **@** character in front.

If yes, all these entries are listed as labels for the entry fields in the **Forms** dialog box. When you have entered the required values and click **OK**, ***DraftBoard*** will substitute the original entries with the **@** characters in front with the text you entered. The original text attributes of these entries such as *style*, *size* or *font* are maintained.

DraftBoard checks not only the layer **TitleBlocks** for @ characters but all selected text objects. All text found between the @ character and the first space will be used as label name. Text in front of the @ character and following the space remains unchanged and not substituted by the text entered into the entry fields in the **TitleBlocks** dialog box. There are more than one variable allowed.

Preparing title blocks for the TitleBlocks command

1. Open one of your own drawing formats.
2. Select the **Text** function from the function palette.
3. Label all entry fields with identifiers such as *Name*, *Date*, etc. or modify existing labels to meet your specifications.
3. Enter in each entry field the label name you put in front of the entry field with an **@** character in front.
4. Assign to all text you entered including the @ character the desired *font*, *style* and *size* using the **Selection** tool from the tool palette.
5. If no layer with the name **TitleBlocks** exists create this layer using the **Layer** command in the **Layout** menu.
6. Select all entries with an **@** character in front using the **Selection** tool.
7. Choose the **Edit objects** command in the **Edit** menu and assign to all selected entries the **TitleBlocks** layer.
8. Save the drawing format.

The @ character you create under Windows using the key combination **Alt+64** or **Alt+064** and on the Macintosh with the key combination **Option+Shift+1**.

Crosshatching and Solid fills

DraftBoard can *crosshatch* and *fill* any enclosed area in your drawing and then automatically update the crosshatching when you change the dimensions of the enclosed area. You can specify a closed area that has a hole or other cutout in it, and ***DraftBoard*** accurately excludes the hole area from the crosshatching and filling.

You can choose the **Fill** command from the keyboard with **Ctrl+B** (*Macintosh:* ⌘**B**) and the **Hatch** command from the keyboard with **Ctrl+H** (*Macintosh:* ⌘**H**).

In the **Pen** menu you will find two fill commands: **Fill** and **Solid Fill** and two crosshatch commands: **Crosshatch** and **Hatch**. **Fill** creates a solid fill for a selection with the current pen color and **Hatch** crosshatches the selection with the current pattern. If you want to change the patterns, choose **Crosshatch** and specify the pattern you want to use. If you want to change the default hatch pattern, make changes to the *preferences file* as described in *The Basics* chapter.

Crosshatch Patterns

In the *Appendix* of this manual you find a representation of all available hatch patterns.

DraftBoard offers a multitude of *DIN* and *ANSI hatch patterns*, of which you can modify the *spacing* and *angle*. You will find these hatches organized in groups such as *metals*, *fluids* etc. in the **Hatch** dialog box.

Adding and specifying Hatch Patterns

In addition to the hatch patterns shipping with ***DraftBoard*** you can use all hatch patterns that are compatible with the hatch format **.pat** of AutoCAD©. When you copy these hatch files into the ***DraftBoard*** folder **DraftBoard\Scripts\VSG\Hatches** the file name is automatically displayed in the **Crosshatch** dialog box. If the file contains more than one hatch pattern the name is followed by an ellipsis **(...)**. When you double-click the file name all patterns contained in the file are displayed.

How to define your own hatch patterns is described under *Hatches* in the *Appendix* of this manual.

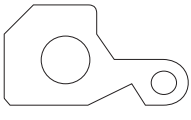
Hatch and Fill boundaries

Before you can *hatch* or *fill* an area you must select the *hatch* or *fill boundaries*. You can select the boundaries with the **Selection** tool or the **Tracer**.

Specifying the Fill or Crosshatch Area

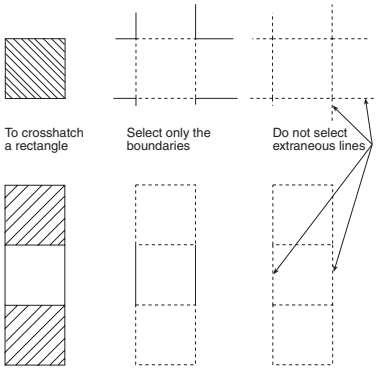
The best way to select a hatch boundary is with the **Tracer** tool.

The selected objects must completely enclose the area you want to crosshatch or fill. For example, to crosshatch the object below, you would select all of the lines making up the part, including the holes.



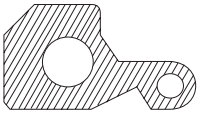
*It is important that you do not select extra objects when you specify the crosshatch or fill area because **DraftBoard** does not know how to treat the extra objects.*

The exterior lines define the boundary for the crosshatching or solid fill. The two holes tell ***DraftBoard*** that those enclosed areas within the original area should not be crosshatched or filled.



Hatching and Filling

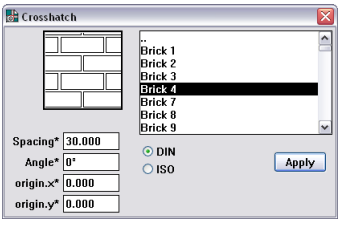
Once you have selected the lines and holes, you can choose the **Hatch**, **Fill**, **Crosshatch** or the **Fill** command from the **Pen** menu and specify the pattern you want to use.



Crosshatch

Using this command you can select a hatch pattern. You can change the default hatch pattern by saving the changes with the **Save Preferences** command in the **Layout** menu under **Preferences**.

Choosing the **Crosshatch** command the following dialog box is displayed.



The **Crosshatch** dialog box allows the following settings:

Patterns In the **Crosshatch** list you select a pattern. Depending which option is checked **DIN** or **ISO** the **Crosshatch** list may contain subdirectories with additional hatch patterns, that you can open with a mouse double-click.

How to add and define your own hatch pattern is described at the end of *Chapter 4* (Preferences) and in the *Appendix* of this manual.

Spacing In this entry field you can set the *spacing* of the hatch pattern. If you don't add a unit that unit is valid that you specified under **Units** in the **Preferences** submenu of the **Layout** menu.

Angle In this entry field you can enter a value for the *angle* of the hatch pattern.

Origin X/Y* In these entry fields you specify values for the **X-** and **Y-Coordinate** of the *Hatch* ***Origin***. You can define these values as well by ***dragging*** the mouse on the ***DraftBoard*** *drawing area*. You just must click in one of the two entry fields and then on the intended point on the drawing area. As soon you release the mouse button, the corresponding values are automatically filled into the entry boxes.

Pattern box The **Pattern** box shows the revised pattern exactly as it will be when the specified object is crosshatched. If you have scaled a drawing, double click in the pattern box to adjust the displayed pattern.

Apply When you click **Apply** all selected and all future objects will be hatched with the settings you specified.

Adding crosshatching

1. Select the objects which represent a closed boundary. The closed boundary may include cutout areas such as a hole. Select both the outside boundary and the hole.
2. Choose **Crosshatch** from the **Pen** menu.
3. Select the pattern you want from the crosshatch list.
   The pattern you select appears in the **Pattern** display area as it will appear on the drawing.
4. Change the spacing and angle, if you want.
   As you make changes to the angle or spacing, the pattern box shows the revised pattern exactly as it will be when the specified object is crosshatched.
5. Click **Apply**.

To crosshatch a figure that is not a closed figure, you must add lines or segment lines to make a closed figure. In this way, you can crosshatch individual sections of the geometry. See *Crosshatching* in the *Advanced Features* section of the Tutorial for a step-by-step example.

Specifying a Hatch Origin

1. Select the objects which represent a closed boundary. The closed boundary may include cutout areas such as a hole. Select both the outside boundary and the hole.
2. Choose **Crosshatch** from the **Pen** menu.

3. Select the pattern you want from the crosshatch list (for example **Bricks 8** in the folder **Extended**).
4. Change the spacing and angle, if you want.

   As you make changes to the angle or spacing, the pattern box shows the revised pattern exactly as it will be when the specified object is crosshatched.
4. Specify in the two entry fields **Origin X*** and **Origin Y*** a value for the new origin of the hatch pattern. Alternatively you can click first in one of the two fields and then in the lower left corner of the hatch boundary on ***DraftBoard*** *drawing area* for the new origin of the selected hatch pattern. The values are automatically filled in into the two **Origin** fields.
5. Click **Apply**.

   The origin of the hatch pattern is moved accordingly.

Editing an existing crosshatch pattern

You can edit hatch patterns with the **Edit Objects** command in the **Edit** menu.

1. Select the existing crosshatching by clicking it with the **Selection** tool.
2. Choose **Crosshatch** from the **Pen** menu.
3. Make the changes you want in the **Crosshatch** dialog box.
4. Click **Apply**.
   The crosshatching changes as you specified.

Hatch Ctrl+H [Macintosh: ⌘H]

This command is located in the **Pen** menu and crosshatches the selected objects using the current crosshatch pattern. If you want to change the crosshatch pattern, choose **Crosshatch** and make your selection from the dialog box.

Crosshatching with the current pattern

1. Select the object you want to crosshatch.
2. Choose **Hatch** from the **Pen** menu.
   The selected area is crosshatched with the current pattern.

Solid Fill

This command in the **Pen** menu fills the selection with the *current pen color* and the *fill tolerances* you specified.

Creating a Solid Fill

The best way to select a fill boundary is with the **Tracer** tool.

1. Select the objects which represent a closed boundary. The closed boundary may include cutout areas such as a hole. Select both the outside boundary and the hole.
2. Choose **Solid Fill** from the **Pen** menu.
   The **Solid Fill** dialog box displays:

3. Set a **Tolerance** for the fill.

   The tolerance controls how smooth the linearization is for fills in *circles*, *arcs*, *ellipses* and closed *splines*. Increasing the value of **Tolerance** makes the fill smoother. The default value is **0.01**. (A value of **0.001** will produce smoother edges than **.01**. This is also effected by Drawing Scale.)

A solid filled object will only cover permanently its background if it is brought to front with the **Bring to front** command in the **Arrange** menu.

4. Click **Apply**.
   The selected object is filled with the current pen color.

Changing the color of an existing solid fill

To change the tolerance of an existing fill, first select the fill by clicking on it. Choose **Solid Fill** from the **Pen** menu. Change the **Tolerance** and press **Apply**. You can also change the **Tolerance in the** Edit Objects dialog box.

1. Select the existing solid fill by clicking it with the **Selection** tool.
2. Choose a new color from the **Color** pop-up menu on the **Pen** menu.

   The **Fill** color changes as you specified and will be visible as soon you deselect the solid fill.

Fill **Ctrl+B** [Macintosh: ⌘H]

This command is located in the **Pen** menu and fills the selected objects using the *current pen color* and *tolerance*. If you want to change the *tolerance*, choose **Solid Fill** and make your selection from the dialog box.

Solid filling with the current pen color and tolerance

1. Select the object you want to fill.
2. Choose **Fill** from the **Pen** menu.

   The selected area is filled with the current pattern and tolerance.

Adding Crosshatch Patterns

DraftBoard offers a wide variety of *DIN* and *ANSI hatch patterns* with editable pattern *spacing* and *angle*. You will find these hatches organized in groups such as *metals*, *fluids* etc. in the **Hatch** dialog box.

A graphical compilation of all crosshatch patterns you find in the *Appendix* of this manual.

Adding Hatch Patterns

All hatch patterns that are compatible with the hatch format **.pat** of AutoCAD© can be used in ***DraftBoard***.

You only must copy hatch files with the *.pat file extension into the ***DraftBoard*** folder **DraftBoard\Scripts\VSG\Hatches**. The file name is then automatically displayed in the ***DraftBoard*** **Crosshatch** dialog box. If the file contains more than one hatch pattern, a folder with this file name is created, that contains all hatchpatterns stored in the ***.pat** file.

Specifying Crosshatch Patterns

You specify your own hatch patterns by creating and binding in *hatch description files* into ***DraftBoard***.

Hatch patterns are defined by *hatch pattern lines,* that are running through a defined origin and are endless repeated at a defined offset distance.

Hatch pattern lines are infinitely long. Each *Hatch pattern line* may contain *single pattern elements* to create dashed or dotted pattern lines. A *Crosshatch pattern* has as well an infinite size and a hatched geometry shows only part of this pattern like a detail window.

Creating Hatch description files

To bind in your own hatch pattern into ***DraftBoard*** you must save the *Hatch description files* as ***ASCII*** *files* into the **DraftBoard** folder

..\Scripts\ VSG\ Hatches

A *Hatch description file* can contain the description of one or several crosshatch patterns.

The file name of a *Hatch description file* is displayed in the **Crosshatch** dialog box of ***DraftBoard***, if it is copied in the folder mentioned above. The file extension of this ***ASCII*** *file* must be always

***.pat**

Without this file extension ***DraftBoard*** is not able to identify this file and cannot display the crosshatch patterns described in this file.

To create a *Hatch description files* you can use any *Text Editor* such as ***NotePad*** for *Windows* or ***SimpleText*** for the *Macintosh*.

Pattern Definitions

A *Hatch description file* contains several sections. The following example explains the file structure describing a simple pattern at a specific angle:

```
;;
;;  DraftBoard
;;  Definition Angle Hatch Pattern
;;
*ANGLE, Angle steel
0, 0,0, 0,.275, .2,-.075
90, 0,0, 0,.275, .2,-.07
```

Each file starts with a *Comment line*, that is indicated by a semicolon (**;**) before each comment.

All lines starting with a semicolon are ignored by ***DraftBoard*** when interpreting the pattern description.

The Comment line is followed by the *Header line* that starts with an asterisk (*****) followed by the pattern *name* and a pattern *description*. Name and Description must be separated by a *comma* (**,**)or a *space* ( ).

The **Crosshatch** dialog box of ***DraftBoard*** shows only the *name* not the *description*.

The name must be unique in the file. If it isn't ***DraftBoard*** uses the first pattern it finds by that name.

The length of the *Header line* is limited to a maximum of 80 characters for *Name*, *Comma* and *Description*.

The header line is followed by the *Hatch data*. Every line of data uses the same format:

angle, x-origin, y-origin, x-offset, y-offset, [, dash1, ...]

Angle defines the angle that this line of hatch pattern data is displayed at. **O** means that the hatch line is drawn horizontally. **90** means that the line is drawn vertically.

Comma separates the numbers.

x-origin The first line of the hatch pattern passes through this x-coordinate.

y-origin The first line of the hatch pattern passes through this y-coordinate.

x-offset defines the distance between line segments; the gap distance. You only use this parameter to specify the offset for vertical or diagonal lines (To specify the distance between dashes, use the **dash1** parameter. In most hatch patterns, the *x-offset* has a value of **0.0**; however, even though this parameter is rarely used, it is not optional.

y-offset defines the vertical distance between repeating lines; this parameter is used by every hatch pattern.

Dash1 You can use one or more **dash1** parameters to define dashes in the hatch pattern line. The **dash1** parameter is optional and not required for solid lines. A positive number, such as **.25** defines the length of a dash. A dot drawn by the hatch pattern is defined by the value **0**. But to be sure that dots will be printed by any type of printer it is recommended to define for dots a very short **dash1** parameters close to **0**.

Tips on creating Pattern Codes

Each line of code applies to a single pattern segment. The two lines of data in our example represent a hatch pattern with two lines.

There is no limit to the number of data lines for hatch pattern definitions. But complex patterns can take a long time to draw on the screen.

You cannot specify a weight or width for hatch pattern lines. The work around is to define two or more very closely spaced lines.

A pattern line consists of straight lines and dots. You cannot specify arcs, circles and other round elements in a hatch pattern file. To simulate circular elements, use a series of very short dashes.

To make it easier to read your hatch pattern code use tabs and spaces to format the code into nice columns like this:

```
Angle x,y-origin  x,y-offset   dash codes
37.5, 0,0,        1.123,1.567  1,-0.5,2,-0.5
```

The hatch description file

Extended.pat

shipping can be modified for the creation of your own patterns.

You find this file in the ***DraftBoard*** folder

..\Scripts\ VSG\ Hatches

CHAPTER

8

Dimensions

Associative Dimensions

Dimension Tools

Dimension Settings

Dimension Attributes

Parametric Dimensions

GD&T

Surface Symbols

Welding Symbols

Dimensions

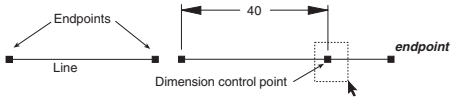
The **Dimension** menu contains commands for displaying the palette of dimensioning tools and for all dimension settings.

Associative Dimensions

DraftBoard's geometric dimensions are associative—when you make a change to the geometry, the dimension changes also.

This associativity is a tremendous time saver because dimensions automatically update whenever you make a change in the geometry. You can even change the units from *English* to *Metric* (one setting in the **Preferences** submenu in the **Layout** menu) and every dimension on your drawing will reflect the change.

When you extend a line by selecting the ***endpoint*** of the line and ***dragging*** it to a new position the dimension changes also, because the dimension has a *control point* at the same position of the line ***endpoint***. So when you select the ***endpoint*** of the line you also select the dimension *control point*.



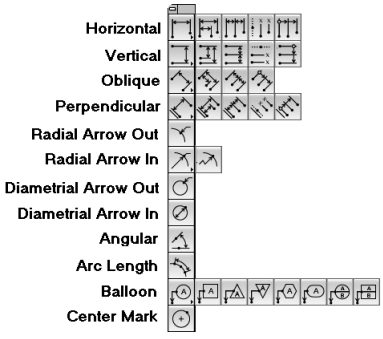
Dimensions are associative relative to the points they measure. If you change the length of a line using **Edit Objects**, the dimension will not update because the point was not changed. To correct the dimension, select the dimension's ***vertex*** point and ***drag*** it to the new ***endpoint*** of the line.

Using the Dimension Tools

DraftBoard's dimensioning tools are on a separate palette which you can ***drag*** around the drawing area. You select dimension tools in the same way you select tools from the tool palette. The first four tools have subpalettes which also work like the subpalettes of the tool palette.

Show Palette

This command in the **Dimension** menu displays the dimension palette on screen.



The dimensioning tools can be used to measure either an object or the area between objects. Once you have dimensioned an object (or space), the dimensions update when you make changes to the geometry.

The first four tools on the dimension palette have subpalettes for base line and chain style dimensions.

Some of the dimensioning tools, such as the *horizontal* and *vertical* tools, require that two points be selected; others, such as the *Radial* and *Diametral*, require only one.

Dimensioning objects and space

1. If necessary, display the **dimension** palette.
2. Select the appropriate dimension tool.

   The dimension pointer is a smart pointer with a *hot spot*—the dot shows which side of the object to select first. If you select as indicated by the position of the dot, the text appears above or to the right of the leader. If you select in the opposite order, the text appears below or to the left of the leader.
3. Click the points you want to measure.

   All settings for dimensions can be set in the *dimension editor* under **Preferences** in the **Dimension** menu.

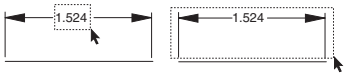
Steps **#4** and **#5** must be done in order. If you change the entry in the text box and then press **Enter**, you can no longer move the dimension, because it will no longer be selected. If you have this problem, select the dimension and ***drag*** it to the location you want.

4. Move the dimension to a new location if its placement is not where you want it.
5. If you are planning to use the dimension with Parametrics, change the entry in the text box on the status line.

Repositioning the dimension

When the dimension text appears, it is selected so that you can move it to a new location. Move the pointer to the dimension text. When the pointer changes to the **Move** symbol, ***drag*** the dimension to its new location.

If you want to move a dimension later, use the **Selection** tool. Click the dimension once, then ***drag*** it to the new location. You can also ***drag*** to select the dimension text or to select the entire dimension or several dimensions at once.



Using the dimension status boxes

The status boxes shown below appear when a dimensioning tool is selected.

Text The **#** symbol in the *text box* indicates that the dimension is the actual value of the object's measurement. If you delete this symbol, the dimension will no longer be associative (that is, it won't update if you change the geometry it measures).

If you are going to use *Parametrics*, you must delete the **#** symbol and enter values, algebraic expressions, or variables in the text box. Parametrics are described in the *Parametrics* chapter of this manual.

Some of the dimensioning tools, such as the **Radial** tools, add a letter in the text status box which appears in the dimension itself. You can add text before or after the **#** symbol. You can enter parentheses to enclose the resulting dimension in parentheses (**#**), or enter **# 2 Plcs** to display the dimension followed by the text **2 Plcs**. When you include the **#** symbol, ***DraftBoard*** displays the measurement of the geometry as well as the text you add.

See as well *Tolerances* in this chapter.

Tolerance The **Upper** and **Lower** boxes appear if you have specified a tolerance format in the *dimension editor* (under **Preferences** in the **Dimension** menu). You enter the limits for the tolerance in these boxes.

Placing dimensions automatically on a separate layer

1. Choose **Preferences** in the **Dimension** menu.

For the ***DIN Standard*** the **Dimension** layer is the default layer.

2. Select in the **Layer** *pop-up* of the *dimension editor* a layer that all dimensions should be placed on.
3. Close the *dimension editor* with a double-click on the **Control** menu (*Macintosh:* ***Close box***).
4. Dimension as usual.

   Subsequent dimensions automatically go on the dimension layer rather than the work layer.

Adding tolerance and limits

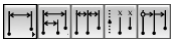
1. Choose the desired style in the *dimension editor* (under **Preferences** in the **Dimension** menu).
2. Dimension as usual.
3. Enter values for the *upper* and *lower* tolerance limits in the status line and press **Enter**. It is also allowed to delete the default values and keep these fields empty or replace the values by text entries. The dimension will display always the content of the status line.

Using fixed dimensions

If you delete the **#** symbol in the text box (on the status line for a newly-created dimension or in the **Edit Objects** dialog box) and replace it with other text, the entry in the text box is fixed—it does not update if you change the units or the size of the geometry.

When object geometry contains *constant dimensions* instead of *actual* measurements, the object geometry is (when selected together with the dimensions) automatically adjusted when using the **Resolve** command in the **Edit** menu.

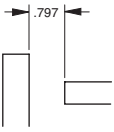
Horizontal Dimension Tools



These tools measure horizontal spaces or the distance between linear objects.

Horizontal Dimension Tool

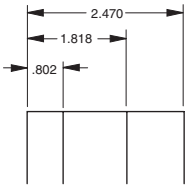
This tool dimensions an object or space horizontally. Click the left point first, then click the right point. (Click the points in the opposite order if you want the dimension to display below the objects.)



When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

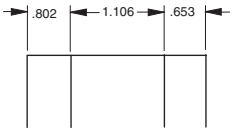
Horizontal Base Line Dimension Tool

This tool dimensions objects or spaces horizontally from a base point.



Horizontal Chain Dimension Tool

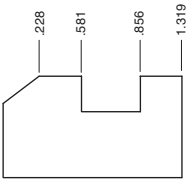
This tool dimensions objects or spaces from end-to-end, horizontally.



Horizontal Ordinate Dimension Tool

This tool dimensions objects or spaces horizontally from a base point.

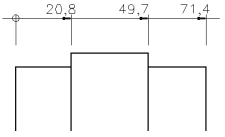
***Remember**: to display the dimension number of the base point you have to click it twice: at the **first** and the **last** dimension points.*



If you want to display the dimension number for the base point you have to click the base point again after you have clicked all other dimension points.

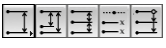
Horizontal Base Chain Dimension Tool

This tool creates a *horizontal* base chain dimension, displaying the rising distance between objects and spaces to a base point.



When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

Vertical Dimension Tools



These tools measure vertical space or the distance between vertical objects.

Vertical Dimension Tool

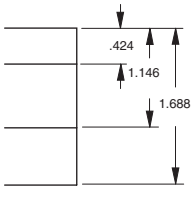
This tool dimensions an object or space vertically. Click the top point first, then click the bottom. (Click the points in the opposite order if you want the dimension to display to the left of the objects.)

When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

Vertical Base Line Dimension Tool

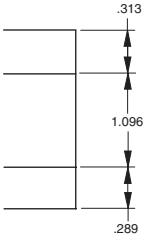
This tool dimensions objects or space vertically from a base point.

If you want to display the dimension number for the base point you have to click the base point **again** after you have clicked all other dimension points.



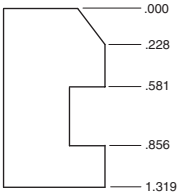
Vertical Chain Dimension Tool

This tool dimensions objects or space from end to end, vertically.



Vertical Ordinate Dimension Tool

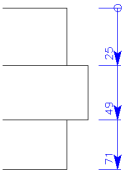
This tool dimensions objects or space vertically from a base point.



If you want to display the dimension number for the base point you have to click the base point **again** after you have clicked all other dimension points.

Vertical Base Chain Dimension Tool

This tool creates a *vertical* base chain dimension, displaying the rising distance between objects and spaces to a base point.



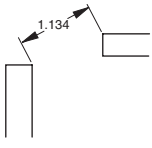
When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

Oblique Dimension Tools

These tools measure space or objects, obliquely from point to point.

Oblique Dimension Tool

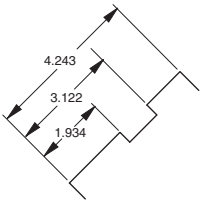
This tool dimensions an object or space from point to point obliquely. Click the left point first, then click the right. (Click the points in the opposite order if you want the dimension to display below the objects.)



When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

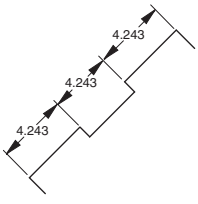
Oblique Base Line Dimension Tool

This tool dimensions objects and spaces point to point obliquely from a base point. ***Note****: The results shown reflect colinear dimension points.*



Oblique Chain Dimension Tool

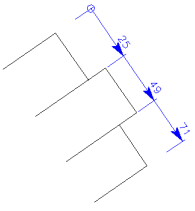
This tool dimensions objects and spaces oblique point-to-point, end-to-end.



Note: The results shown reflect colinear dimension points. Lines are *colinear* if they have a common invisible *carrier line*. ***DraftBoard*** can recognize colinear lines only if they touch at least in one endpoint.

Oblique Base Chain Dimension Tool

This tool creates a *oblique* base chain dimension, displaying the rising distance between objects and spaces to a base point.

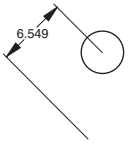


Perpendicular Dimension Tools

These tools measure an object or space perpendicular to another line. If most of your dimensions start from a line edge, you can use this tool to generate both horizontal and vertical dimensions.

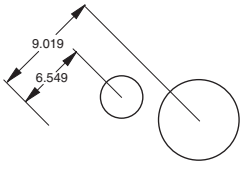
Perpendicular Dimension Tool

This tool dimensions an object or space perpendicular to a line. Click the base line first, then click the object or location.



Perpendicular Base Line Dimension Tool

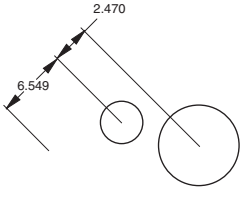
This tool dimensions between a point or object perpendicular to an existing base line.



Note: Be certain that you click on the base line, not on the endpoint of the base line. When the dimension text appears, you can **drag** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

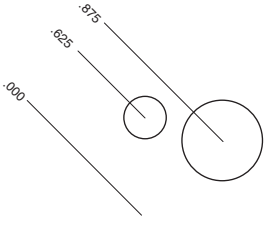
Perpendicular Chain Tool

This tool dimensions between points or objects perpendicular to an existing baseline.



Perpendicular Ordinate Dimension Tool

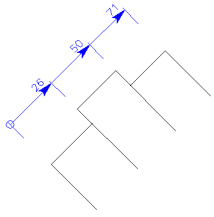
This tool dimensions between a point or object perpendicular to an existing base line.



If you want to display the dimension number for the base point you have to click the base point **again** after you have clicked all other dimension points.

Perpendicular Base Chain Dimension Tool

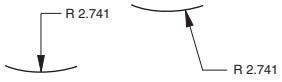
This tool creates a *oblique* base chain dimension, displaying the rising distance between objects and spaces to a base point.



Radial Arrow Out Dimension Tool

This tool measures the radius of a circle, arc, or fillet with the arrow on the side of the arc indicated when the arc is picked. To dimension the radius of a circle, arc, or fillet click near it. If you click outside it, the dimension text appears outside, and if you click inside, it appears inside.

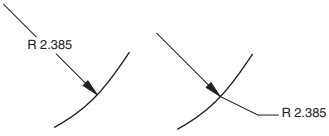
You can move a dimension arrow to a new location by ***dragging*** a selection marquee around the *control point* at the top of the *dimension arrow* and then ***drag*** it to a new location.



When the dimension appears the leader line is placed at the nearest ***15°*** *increment* from the location you clicked. You can ***drag*** the text to a new location.

Radial Arrow In Dimension Tool

This tool measures the radius of a circle, arc, or fillet with the arrow inside the geometry. To dimension the radius of a circle, arc, or fillet, click near it. If you click outside it, the dimension text appears outside, and if you click inside, it appears inside. Either way the arrow line starts from the arc center.

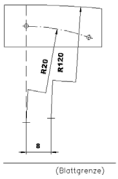


When the dimension appears the leader line is placed at the nearest ***15°*** *increment* from the location you clicked. You can ***drag*** the text to a new location.

Extended Radial Arrow In Dimension Tool

This tool measures the radius of a *circle*, *arc*, or *fillet* with the arrow inside the geometry. To dimension the radius of a *circle*, *arc* or *fillet*, click near it (the *Drawing Assistant* must show ***on***). If you click outside it, the dimension text appears outside, and if you click inside, it appears inside. Either way the arrow line starts from the arc center.

In contrast to the two other radial dimensions the *Extended Radial dimension* allows to draw the dimension line of a large radius shortened and angled. Both, the part of the dimension line with the dimension arrow and the angled part of the dimension line, point to the geometrical center.



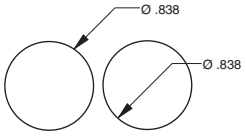
When the dimension appears the leader line is placed at the nearest ***1°*** *increment* from the location you clicked.

The leader line is always drawn to the geometrical center. To shorten and angle the leader line, you must first ***drag*** the dimension text to a new location and select then the end of the dimension line at the geometrical center and ***drag*** it away from the center point.

The dimension line is shortened and angled in a z-Form, where both legs point to the geometrical center.

Diametral Single Arrowhead Dimension Tool

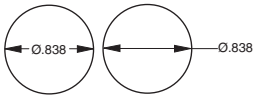
This tool measures the diameter of a circle. To dimension the diameter of a circle, click near the circle. If you click outside the circle, the dimension text appears outside, and if you click inside, it appears inside.



When the dimension appears the leader line is placed at the nearest ***15°*** *increment* from the location you clicked. You can ***drag*** the text to a new location.

Diametral Double Arrowhead Dimension Tool

This tool measures the diameter of a circle. To dimension the diameter of a circle, click near the circle. If you click outside the circle, the dimension appears outside, and if you click inside, it appears inside.

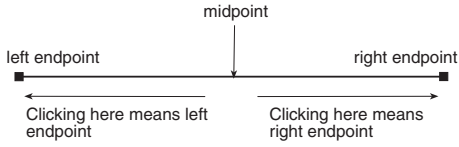


When the dimension appears the leader line is placed at the nearest ***15°*** *increment* from the location you clicked. You can ***drag*** the text to a new location.

Angular Dimension Tool

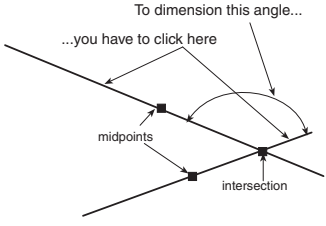
This tool measures the angle between two lines. To dimension an angle, click each line near the endpoint from which you want the angle measured. The inside angle is measured between the sides with the extension lines extending from the endpoints nearest the location you clicked on the line. The smaller angle is used.

The angle between the endpoints of the lines is measured so you do not have to click these endpoints, just near them. ***DraftBoard*** interprets
all mouse clicks left of the line midpoint as the left endpoint and all mouse clicks right of the line midpoint as the right endpoint.



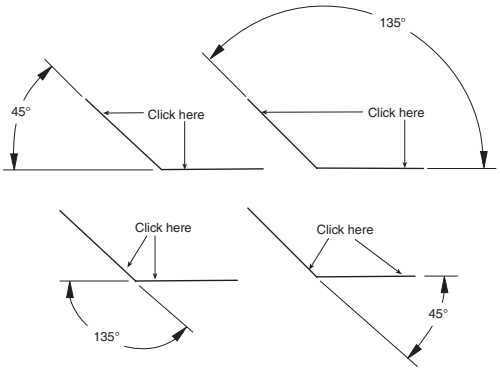
If you dimension intersecting lines don't mix up the intersection with the midpoint of the lines.

Angular dimensions will display all angles properly if you keep the angles larger than **3** degrees. For dimensions smaller than that, you should build the dimensions manually using lines and text objects.



The Angular **Dimension** tool will not dimension angles over 180°.

If the lines don't meet, the angle is measured from the invisible extension of the lines. The extension appears as part of the dimension.



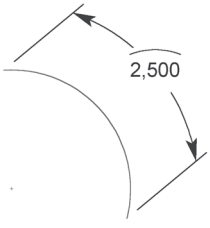
When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

Arc Length Dimension Tool

This tool measures the length of an arc. To use it, you have to move the cursor to the arc to be measured and click on the following points:

- one end of the arc,
- the middle of the arc and
- the opposite end of the arc (as if you were creating a three-point arc).

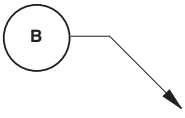
The **Arc Length Dimension** tool does not put witness lines perpendicular to the arc for angles less than **180** degrees.



Like other dimensions you can manipulate the text by changing its size or font using the *Dimension Editor*. You can move the text by ***dragging*** it to the desired position.

Balloon Symbol

This tool attaches a *Balloon symbol* to geometry, placed in the location you click.



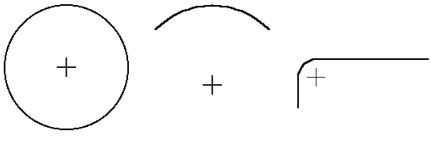
All *Balloon Symbols* use as line style **Balloon**. Therefore you should use this line style as well when creating *Balloon Symbols* manually .

You can enter text and the diameter of the balloon in the Status Line.

Center Mark

This tool marks the ***center*** of a *circle*, *arc* or *fillet*.

To mark the ***center*** of a *circle*, *arc* or *fillet*, click near it (the *Drawing Assistant* must show ***on***).



Dimension Settings

Scaling a drawing using the **Drawing Size** or **Sheet Into View** command affects always the *dimension appearance* regardless of the specified *dimension settings*.

Dimension Appearance

When scaling a drawing all *existing dimensions* are scaled as well. Only dimensions placed after scaling a drawing are displayed at their specified text size and are not adjusted to the current *view scale*.

Dimensions are always displayed correctly at their true size, when:

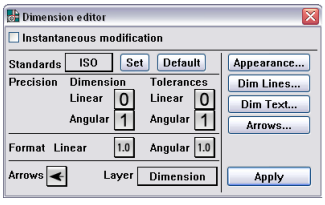
- the option *keep dimension text size* in the **Drawing Size** dialog box is checked while scaling
- a drawing is dimensioned **after** scaling
- or the original text size is set **after** scaling.

A detailed description about Scaling you find in ***Chapter 9*** *Basic Drafting* of this manual.

Dimension Editor

All attributes for dimensions created with the dimension tools are set with the *Dimension Editor*.

Choosing the **Preferences** command in the **Dimension** menu displays the *Dimension Editor*:

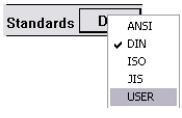


The *Dimension Editor* allows the following settings:

Standards

When you select a new standard the *dimension editor* displays the related settings.

The **Standards** *pop-up* menu allows you to select one of the following dimension standards:



You can save the selected *standard* permanently with the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

All standards can be modified. The **Set** button saves all modifications made in the *Dimension Editor* to the current standard. These changes are valid until the end of the current ***DraftBoard*** session or until you modify the settings again. You can save the selected *standard* permanently using the **Save Preferences** command from the **Preferences** submenu in the **Layout** menu.

Setting a dimension standard

1. Choose **Preferences** in the **Dimension** menu.
   The *Dimension Editor* is displayed.
2. In the **Standards** *pop-up* menu ***drag*** to the standard you want to modify.
3. Modify the desired parameters in the *Dimension Editor*.
4. Click the **Set** button.
   All settings are valid until the end of the current ***DraftBoard*** session or the settings are modified again.

All modifications made are only valid for the current standard. When you want to edit only some selected dimensions you must select the current standard again after you clicked the **Apply** button.

5. Click the **Apply** button if you want to assign the new settings to selected dimensions.
   All selected dimensions are redrawn and reflect the new appearance set in the *dimension editor*.
6. Close the *Dimension Editor* by clicking the **Close** box in the upper left corner of the *Dimension Editor*.

Restoring a dimension standard

1. Choose **Preferences** in the **Dimension** menu.
   The *dimension editor* is displayed.
2. In the **Standards** *pop-up* menu ***drag*** to the standard you want to restore to its default settings.

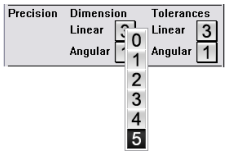
3. Click the **Default** button.
   All settings of the selected standard are restored to its default values.
4. Close the *Dimension Editor* by clicking the **Close** box in the upper left corner of the *Dimension Editor*.

Standards and modifications

All modifications are assigned to the current standard. When you want to edit only some selected dimensions you have to select the standard again after you have clicked the **Apply** button to restore its original settings.

Precision

The **Precision** *pop-up* menus in the *Dimension Editor* allows you to set the number of decimal places for *dimensions* and *tolerances*.



The **Precision** of *angular dimensions* can be set to *degrees*, *minutes*, and *seconds*.

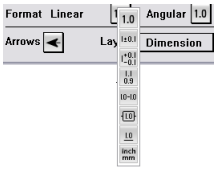
Set the Precision

1. Choose **Preferences** in the **Dimension** menu.
   The *Dimension Editor* is displayed.
2. Set the number of decimal places for *linear* and *angular dimensions* and *tolerances* in the corresponding *pop-up* menus.
3. Click the **Apply** button.
   The specified precisions are valid for all selected and all future dimensions until the end of the current ***DraftBoard*** session.

You can save the selected *precision* permanently with the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

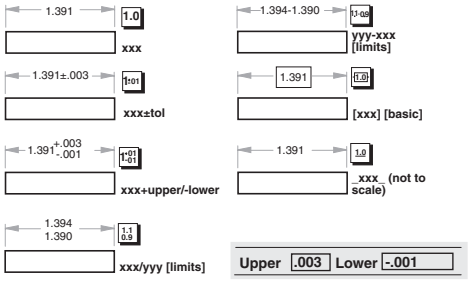
Dimension Formats

Dimension formats including tolerance formats can be specified in the main window of the *Dimension Editor*.



Adding Tolerance Formats

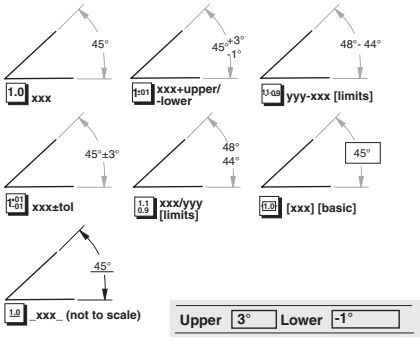
1. Choose **Preferences** in the **Dimension** menu.
   The *Dimension Editor* is displayed.
2. Select *Dimension formats* for *linear* and *angular* dimensions.
   The illustration below shows the appearance of each *Dimension format* for *linear dimensions* if you entered the limits shown in the status line.



GD&T Dimensions use the **GD&T** Format.

The illustration below shows the appearance of each *tolerance format* for *angular dimensions* if you entered the limits shown in the status line.

GD&T Dimensions use the **GD&T** Format.



3. Click the **Dim Text** button.

   The **Dimension Text** dialog box is displayed.

The default value for the *DIN standard* is **70%** of the dimension text size.

4. Enter a value for the *tolerance text size* into the **Tol.-Text size** field. This value specifies the *text size for tolerances* in percentage of the *dimension text size*.
5. Click **OK**.

   The **Dimension Text** dialog box is closed.
6. Close the *Dimension Editor* by clicking the **Close** box in the upper left corner of the *Dimension Editor*.
7. Draw a line.
8. Select a linear dimension from the *Dimension Tool palette* such as the **Horizontal Dimension** tool and dimension the line.
9. Enter in the status line values for the *upper* and *lower* tolerance limits.
10. Press the **Enter** key.

    The dimension is redrawn and displays the tolerance limits you specified.

    The specified *tolerance limits* become the default limits until you enter different values while dimensioning.

8

Entering Text into the Tolerance fields

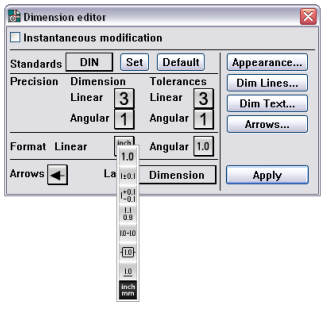
Instead of typing in numbers to calculate the tolerances you can also enter text, such as **H7** in the *upper* and *lower* entry fields of the status line or even leave one of these fields empty. The dimension will reflect exactly what you have entered in the status line.

You must press the **Enter** key to activate the entries made in the Status Line.

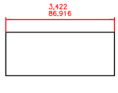
When you enter text in the status line you should use the **upper/ -lower** tolerance format in the *Dimension Editor*.

Dual Dimension Format

For dual dimensions (displaying simultaneously ***mm*** and ***inch*** *units,* independent from the units set in the **Units** dialog window in the **Preferences** submenu in the **Layout** menu) choose the corresponding format in the format list for linear dimensions.



When you select the dimension format **Dual Dimension** in the *Dimension editor*, all dimensions will display simultaneously ***inch*** and ***mm*** values.



Dimension Layer

DraftBoard places dimensions automatically onto that layer you have selected from the **Layer** *pop-up* menu in the *Dimension Editor*.

Placing dimensions automatically on a separate layer

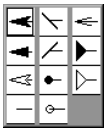
1. Choose **Preferences** in the **Dimension** menu.
2. Select a layer in the **Layer** *pop-up* menu in the *Dimension Editor* where all dimensions should go to.
3. Close the *Dimension Editor* by clicking the **Close** box in the upper right corner of the *Dimension Editor*.
4. Dimension as usual.

   Subsequent dimensions automatically go on the dimension layer rather than the work layer.

For the **DIN** Standard the **Dimension** Layer is set as default layer.

Arrows

The **Arrows** *pop-up* menu allows you to choose one of 11 arrowheads for dimension lines.



Arrowheads for *arrow lines* are defined separately under **Arrows** in the **Pen** menu.

The arrowhead selection for dimensions is valid until the end of the current ***DraftBoard*** session. You can save the selected *arrowhead* permanently with the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

Instantaneous Modification

When this option in the *Dimension Editor* is checked all changes made in the *Dimension Editor* take effect immediately on all selected dimensions. The **Apply** button is disabled as long as this option is active.

Apply button

When you click the **Apply** button all modifications made in the *Dimension Editor* are applied to all selected dimensions except the two values in the **Appearance** dialog that are marked with a **#** symbol in front.

All modifications are valid for future dimensions until the end of the current ***DraftBoard*** session. When you want to edit only some selected dimensions you have to select the standard again after you have clicked the **Apply** button to restore its original settings.

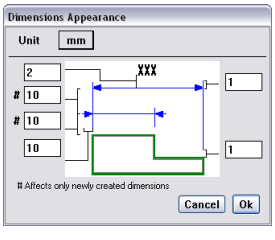
If you want to make these changes permanent you must choose **Save Preferences** in the **Preferences** submenu of the **Layout** menu.

Closing the Dimension Editor

You close the *dimension editor* by clicking the **Close** box in the upper right corner of the *Dimension Editor*. All modifications made in the *Dimension Editor* are valid until the end of the current ***DraftBoard*** session

Appearance

All settings for *Dimension Appearance* are done in the **Appearance** dialog box. When you click the **Appearance** button the following dialog box is displayed:



The **Appearance** dialog box allows the following settings:

Units In this *pop-up* menu you select the units (**mm** or **inch**) for all settings in the **Appearance** dialog box. These units are only valid for the setting in the **Appearance** dialog

box and do not affect the units set for the drawing in the **Preferences** submenu of the **Layout** menu.

Entry fields In these entry boxes you enter the desired offset values for the different elements of a dimension.

When you click the **Apply** button in the *Dimension Editor* the new settings are assigned to the selected dimensions except the two values you entered in the *# fields* which will affect only future dimensions.

#-Symbols Values you enter in the two entry fields which have a *# Symbol* in front affect only newly created dimensions (not selected dimensions). These two entry fields display values for the perpendicular distance between objects being dimensioned and for the *first* and all further *dimension lines*.

OK When you click the **OK** button the **Appearance** dialog box is closed and all settings are assigned to the current standard.

Modifying the dimension appearance

1. Select all dimensions you want to modify. If no dimensions are selected all changes are valid only for future dimensions.
2. Choose **Preferences** in the **Dimension** menu to display the *Dimension Editor*.

If you want to modify the *Appearance* for another *Dimension Standard* you must first change the **Standard** in the *Dimension Editor* and then modify the *Appearance* in the **Appearance** dialog box.

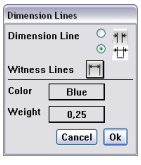
3. ***Drag*** in the **Standard** *pop-up* menu to the standard you want to modify.
4. Click the **Appearance** button in the *Dimension Editor*.
   The **Appearance** dialog box is displayed.
5. Select **mm** or **Inch** in the **Units** *pop-up* menu for all changes you want to make in the **Appearance** dialog box.
6. Modify the *Dimension Appearance* by entering appropriate values in the related entry fields.
7. Click **OK**.
   The **Appearance** dialog box is closed.
8. Click the **Apply** button if you want to assign the new settings to selected dimensions.
   All selected dimensions are redrawn and reflect the new appearance set in the **Appearance** dialog box except the two values you entered in the ***# fields*** which will affect only newly created dimensions.

8 Click the **Standard** button in the *Dimension Editor* if you want to reset all parameters to its original values.

Dimension Lines

All settings for *Dimension Lines* are done in the **Dimension Lines** dialog box.

When you click the **Dim Lines** button the following dialog box is displayed:



The **Dimension Lines** dialog box allows the following settings:

The **Dimension line** options allow you to choose whether the witness lines are connected by a dimension line (DIN) or not for all dimension text that is not placed between witness lines.

Witness lines In this *pop-up* menu you select the number of witness lines: *left side*, *right side*, *both sides* or *no witness lines*.

Color In this *pop-up* menu you select the *color* for the dimension lines.

Weight In this *pop-up* menu you select the *line weight* for dimension lines. You can choose from the same *line weights* offered in the **Weight** submenu of the **Pen** menu.

If you want to make these changes permanent you must choose **Save Preferences** in the **Preferences** submenu of the **Layout** menu.

OK When you click the **OK** button the **Dimension Lines** dialog box is closed and all settings are assigned to the current standard.

Modifying dimension lines

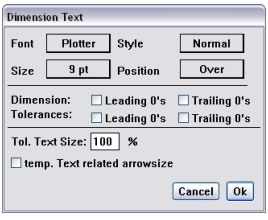
1. Select all dimensions you want to modify. If no dimensions are selected all changes are valid only for future dimensions.
2. Choose **Preferences** in the **Dimension** menu to display the *Dimension Editor*.
3. ***Drag*** in the **Standard** *pop-up* menu to the standard you want to modify.
4. Click the **DIM Lines** button in the *dimension editor*.
   The **Dimension Lines** dialog box is displayed.
5. Select the all options you want to set for dimension lines.
6. Click **OK**.
   The **Dimension Lines** dialog box is closed.
7. Click the **Apply** button if you want to assign the new settings to a selected dimension.
   All selected dimensions are redrawn and reflect the new appearance set in the *dimension editor*.

Dimension Text

All settings for *dimension text* (including *Welding* and *Surface Symbols*) are done in the **Dimension Text** dialog box.

All changes made in the **Text** menu have no effect on dimensions.

When you click the **Dim Text** button the following dialog box is displayed:

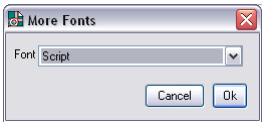


If the option **Instantaneous modifications** in the main window of the *Dimension editor* is checked all modifications done in the **Dimension Text** dialog box take effect immediately.

The **Dimension Text** dialog box allows the following settings:

Font — This *pop-up* menu allows you to choose the *text font* for the selected standard and for all selected and future dimensions.

more — The **Font** submenu can display up to *20 fonts*. If you have installed more than 20 fonts you have to click the **more** button to get access to all available fonts.



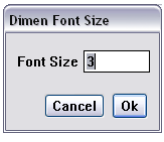
The **Fonts** dialog box lists all fonts installed on your computer including ***DraftBoard's*** plotter fonts.

Whenever you want a font set to exact specifications, use a **Plotter** font, which conforms to ANSI standards, since *PostScript* and *TrueType* fonts are *Proportional Fonts* that can deviate slightly from the specified size.
When you scale a drawing using the **Sheet Into View** or the **Drawing Size** command and having the options *keep text size* and *keep dimension text size* not checked, you have to select all *dimensions* and assign their text size in the *Dimension Editor* again.

Size — This *pop-up* menu allows you to choose the *text size* for the selected standard and for all selected and future dimensions.

The *text size* can be specified either in *point*, in *mm* or *inch* depending on the units you have set under **Units** in the **Preferences** submenu of the **Layout** menu.

Other — Choosing **Other** from the **Size** submenu brings up a dialog box allowing you to specify a nonstandard *text size* in the units you've set for your drawing. You can specify point size by entering **pt** after the value in the entry box; for example, if you enter **11 pt** the font size will be *11 point*.



Style — This *pop-up* menu allows you to choose the *text style* for the selected standard and for all selected and future dimensions.

Certain *Font* ***Styles*** are only available for the *Macintosh*.

Leading 0 Determines whether or not the zero to the left of the decimal point appears.

Trailing 0's Determines whether or not zeros to the right of the decimal point are displayed. If set, the precision determines the number of trailing zeros.

Tol. Text size In this entry field you specify the *text size for tolerances* as a percentage of the *dimension text size*.

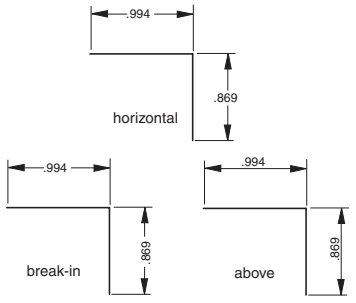
temp. Text related arrow size When this option is checked the ***arrow size*** of the *dimension lines* will be adjusted when the *text size* is modified. The default arrow size for dimension lines is normally only increased or reduced while scaling a drawing. Using very large or small dimension text makes it sometimes necessary to adjust the *arrow size* to the selected *text size*.

This option is deactivated by default and can be used only for selected dimensions using the **Apply** button.

Position This *pop-up* menu allows you to set the *position* of *dimension text* for the selected standard and for all selected and future dimensions. You can choose from the following options:

- *horizontal*
- *over*
- *break-in*
- *under*

The first option determines if the dimension text is always displayed *horizontally*. The following three options determine if the dimension text is placed above (*over*) in between (*break-in*) or below (*under*) the dimension line.



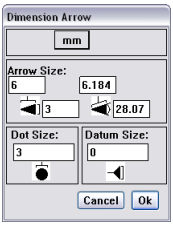
Modifying dimension text

1. Select all dimensions you want to modify. If no dimensions are selected all changes are valid for future dimensions.
2. Choose **Preferences** in the **Dimension** menu to display the *Dimension Editor*.
3. ***Drag*** in the **Standard** pop-up menu to the standard you want to modify.
4. Click the **DIM Text** button in the *Dimension Editor*.
   The **Dimension Text** dialog box is displayed.
5. Select all the options you want to set for dimension text.
6. Click **OK**.
   The **Dimension Text** dialog box is closed.
7. Click the **Apply** button if you want to assign the new settings to a selected dimension.
   All selected dimensions are redrawn and reflect the new appearance set in the *dimension editor*.

Arrow Size

All settings for *Dimension Arrows* are done in the **Arrow Size** dialog box.

When you click the **Arrow Size** button the following dialog box is displayed:



The **Arrows** dialog box allows the following settings:

Units In this *pop-up* menu you select the units (**mm** or **inch**) for all settings in the **Arrows** dialog box. These units are only valid for the setting in the **Arrows** dialog box and do not affect the units you set for your drawing in the **Preferences** submenu in the **Layout** menu.

Arrows Size In these entry boxes you can set the size of the arrowhead either by specifying its *length/height ratio* (ANSI) or setting values for the *length of the edge* and the *angle of the tip* of the arrowhead (DIN).

Dot Size This value describes the diameter of *circular* and *slash* arrowheads.

Setting the Arrow Size

1. Select all dimensions you want to modify. If no dimensions are selected all changes are valid for future dimensions.
2. Choose **Preferences** in the **Dimension** menu to display the *Dimension Editor*.
3. ***Drag*** in the **Standard** *pop-up menu* to the standard you want to modify.
4. Click the **Arrow Size** button in the *Dimension Editor*.
   The **Arrow Size** dialog box is displayed.
5. Select all the options you want to set for the size of the arrowheads.
6. Click **OK**.
   The **Arrow Size** dialog box is closed.
7. Click the **Apply** button if you want to assign the new settings to a selected dimension.
   All selected dimensions are redrawn and reflect the new appearance set in the *dimension editor*.

Editing Dimension Attributes

All dimension attributes such as *dimtextangle*, *dimlineangle*, *dimsublineangle* can be modified by selecting the dimension and editing the attributes in the **Edit Objects** dialog box in the **Edit** menu.

Parametric Dimensions

You must dimension objects to use the parametrics feature; you must specify values, variables, or expressions for each dimension. *Parametrics* is discussed in greater detail in the *Parametrics* chapter.

Creating a parametric dimension

1. Create the dimension as usual.
   The status line displays a **#** symbol in the text box.
2. Enter a value (variable or algebraic expression) in the text box.
   The dimension reflects the value or variable you entered, regardless of the actual value of the geometry you just dimensioned.
3. Continue dimensioning as needed to define the part completely.
4. Use the parametric feature as described in the *Parametrics* chapter.

Converting parametric dimensions to associative dimensions

When you bring in a parameterized part with the **Import** command, the geometry is drawn as you specify, but the dimensions display the variables and expressions which defined them as symbols. You can change the dimensions to be associative so that they reflect the actual measurements of the geometry, even if you change the geometry.

1. Select the dimensions.
2. Choose **Edit Objects** from the **Edit** menu.

GD&T

GD&T stands for **G**eometric **D**imensioning and **T**olerancing, a *DIN ISO 1101* and a *U.S. government* standard. Since this annotation may be new to you, this section is an overview of ***GD&T*** *standards* and includes a brief explanation of the components of a ***GD&T*** *label*, using the true position of holes as an example.

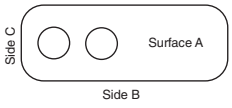
Background

Before World War II, a single vendor did most military manufacturing. However, during the war, it became a matter of national security to diversify weapons manufacturing so a single plant was not the only source of a vital part. When more than one manufacturer interpreted specifications and blueprints, difficulties arose.

When several companies manufacture the same part, each must drill holes within the same tolerance for the parts to be interchangeable. Even though manufacturers thought they were following the tolerance specifications, the parts were not interchangeable. The problem arose from the **order** in which measurements were made.

Alignment Information

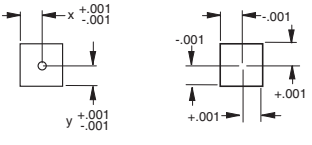
A part was usually laid on a flat surface (**A**) and aligned first with one straight edge (**B**), then with another (**C**).



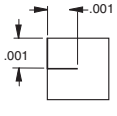
If one manufacturer aligned with edge **B**, then edge **C**, but another aligned with edge **C**, then edge **B**, the resulting measurements would probably be different. The ***GD&T*** *Standards* tell a manufacturer the order of alignment.

Bonus Tolerance

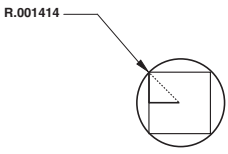
In addition to the alignment information, a manufacturer gets a bonus from ***GD&T*** *Tolerances*. In conventional drafting, the tolerance for hole drilling results in a square target area. For example, examine the plate on the left below. The positional tolerance for the hole is **±.001**. The target area (magnified and shown on the right) is from **.001** left of the perfect center to **.001** right of the perfect center, and **.001** above the perfect center to **.001** below the perfect center.



GD&T provides a circular target area for the hole. First measure from the center in one direction by the distance of the tolerance; then measure the same distance perpendicular to that point.

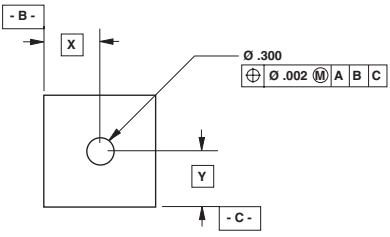


You can then use the distance from the endpoint of the second line to the perfect center as the radius of a circle to create a circular target—the *bonus tolerance*.



In that way, the target area for the hole's center is the area circumscribing the square of standard dimensions.

The standard information (basic dimensions) about the target area appears in a box with a ***GD&T*** *feature control frame*.



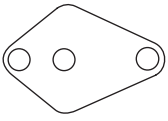
Basic Dimensions

Since the tolerance for a dimension is in the feature control frame, ***GD&T*** dimensions are also basic dimensions and appear in a box. The value defines the theoretically perfect location and implies that the tolerance is in the feature control frame.

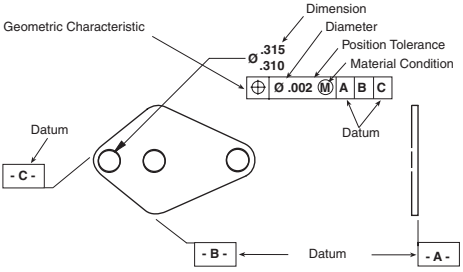
To specify the basic dimension format, set the ***GD&T*** *Format* in the *Dimension Editor*. Choose **Preferences** in the **Dimension** menu to display the *Dimension Editor*

GD&T Feature Control Frame

This section discusses the ***GD&T*** *Labelling* of the left hole in the plate below.



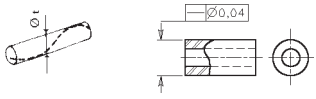
Typically, the ***GD&T*** *Feature Control Frame* appears as shown below:



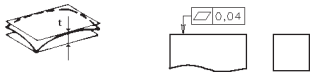
Geometric Characteristic

The first section of the *Feature Control Frame* contains the geometric characteristic symbol. The feature control shown above describes *True Position*—how to locate the circular target area for the hole. The possible characteristics are as follows:

Straightness The axis of an item of revolution or element of a surface is a straight line.

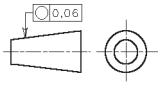


Flatness All elements of a surface in one plane.



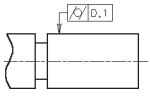
Roundness

All points of a surface of revolution equidistant from an axis that is intersected by a plane perpendicular to that axis.



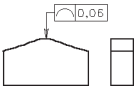
Cylindricity

All points of a surface of revolution equidistant from a common axis.



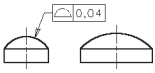
Profile of Line

A 3D element projected to a plane.



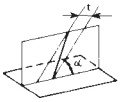
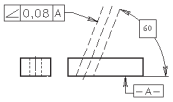
Profile of Surface

A 3D surface projected to a plane.



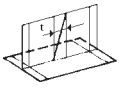
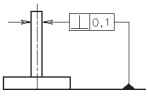
Angularity

An axis or planar surface at a specified angle from another axis or datum plane.

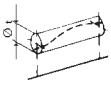
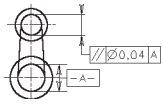
Perpendicularity

An axis or planar surface at a right angle to a datum axis or datum plane.

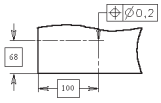
Parallelism

An axis or planar surface equidistant along its length to a datum axis or datum plane.

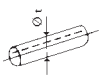
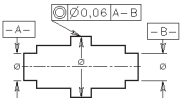
True Position

A zone of tolerance for a center axis or center plane.



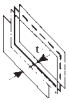
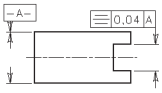
Concentricity

The axes of all elements in a cross-section share a common datum axis.

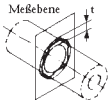
Symmetry

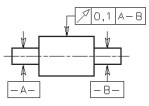
The location of a feature relative to a center plane.

Runout

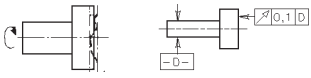
Circular Runout





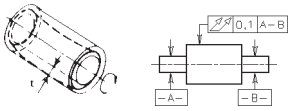
Circular elements of a surface of revolution associated with a datum axis.

Surface Runout



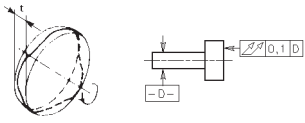
Total Runout

Total Circular Runout



All elements of a surface of revolution associated with a datum axis.

Total Surface Runout



Diameter

The second section of the feature control frame specifies whether the tolerance applies to a circular zone. If it does, you should specify the diameter symbol (**Ø**) from the pop-up menu.

If the diameter symbol is appropriate for the geometry, you should display it. It is usually not used with flatness, roundness, cylindricity, profile of line or surface, angularity, runout or total runout.

Position Tolerance

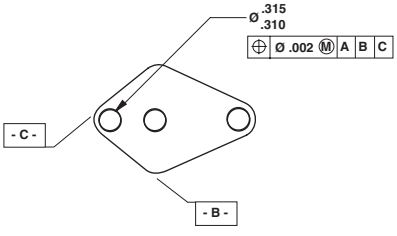
The third section of the feature control frame defines the tolerance allowed under the stated conditions.

Material Condition

The fourth section of the feature control frame provides the material condition for limiting the tolerance. You can use any of three material conditions: *Maximum*, *Least*, and *Regardless* of Feature Size.

Maximum Material Condition

Maximum **M**aterial **C**ondition (**MMC**) indicates that a feature contains the maximum amount of material within the specified tolerance limit. For drilling holes, MMC means that the drill bit is accurate—it drills the smallest hole. For shafts, **MMC** results in the largest dimension.



Maximum **M**aterial **C**ondition is the most commonly used symbol for parts.

In the example above, this material condition symbol means that the tolerance in the *Feature Control Frame* (**.002**) applies to the location of the smallest hole (**.310** in this case).

As the hole size nears its upper limit **(.315**), the location of the hole increases from **.002** to **.007** (the tolerance for the location, **.002**, plus the tolerance for size, **.005**).

Least Material Condition

Least **M**aterial **C**ondition (**LMC**) indicates that a feature contains the minimum amount of material within its specified tolerance limit. For holes, the swept area is the largest and the drill bit drills the largest hole. For shafts, the tolerance is the smallest allowable dimension.

Least **M**aterial **C**ondition is most frequently used to control wall thicknesses of parts, as well as for tool fixtures and inspection gauging.

In the example above, this material condition symbol means that the tolerance (**.002**) applies to the location of the largest hole (**.315**). As the hole size nears its lower limit (**.310**), the tolerance for the location of the hole increases from **.002** to **.007** (the tolerance for the location, **.002**, plus the tolerance for size, **.005**).

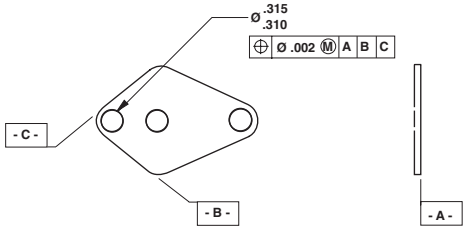
Regardless of Feature Size

The **R**egardless of **F**eature **S**ize (**RFS**) material condition means the tolerance applies regardless of the size of the feature (within the specified tolerance limit). For holes, the location tolerance is the same regardless of whether the hole is the smallest or largest allowable size. For shafts, the roundness tolerance is the same regardless of the shaft diameter.

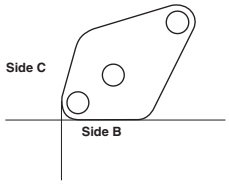
In the example above, this material condition symbol in the feature control frame (**.002**) applies to the location of the hole whether the hole is at its smallest (**.310**) or largest (**.315**). This material condition would control an axis in space, so you probably wouldn't use *Regardless of Feature Size* for this hole.

Datum

The last three sections of the frame show the alignment order to position the part. This is easier to see in a part which does not have perpendicular sides. A typical engineering drawing might appear as shown below:



For proper drilling alignment and measurement, this part would lie on a flat surface (**Surface A**), with **Side B** pushed against the first straight edge, and then **Side C** pushed against the other straight edge.



GD&T tells a manufacturer which surface to align first. If you align the part with **Side C** before **Side B**, the holes do not line up in the same way as they would if you align with **Side B** before **Side C**.

The rule for points of contact per surface is as follows:

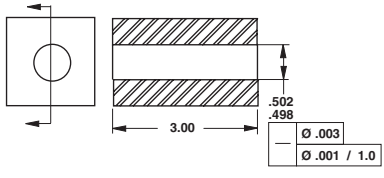
Surface	Points of contact (minimum)
1st	3
2nd	2
3rd	1

Datum Material Condition

When showing a datum in a feature control frame, you also have the option of indicating a material condition. Such a modifier should only be specified for a datum that is an axis or centerplane of a feature, such as a hole or boss. Material Condition modifiers do not apply to planes such as the straight side or flat bottom of a part.

Composite GD&T Symbols

A composite symbol uses more than one line of the feature control frame to specify more than one tolerance for the same feature.

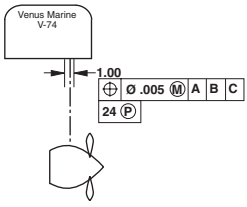


The first line of the feature control frame means that the hole must be straight throughout within a cylindrical tolerance of **.003** diameter for the total of its length, **3.00** inches. The second line of the frame means

that the hole must be straight for any **1.00** inch portion within a cylindrical tolerance of **.001** diameter. ***DraftBoard*** creates the composite frame automatically if the same symbol is in **Line 1** and **Line 2**.

Projected Tolerance Zone

This entry box of the feature control frame specifies an area above the actual part where the tolerance of a feature should still apply if the feature were extended or projected into the area. For example, if an outboard motor has a hole in the housing, the hole has to be straight not only **through** the housing, but also **down** into the propeller assembly, so that the drive shaft aligns properly.



In the motor example, the tolerance for the hole should be checked ***4*** *units* from the actual hole.

The GD&T Label

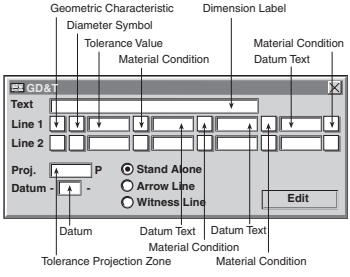
Feature control frames are created with **GD&T** command in the **Dimension** menu.

GD&T

This command in the **Dimension** menu creates a label—a feature control frame for showing *dimensions*, *alignment*, and *tolerances*. The dialog box looks like a complex **GD&T** symbol into which you may enter the information you need.

The ***GD&T*** *label* uses the plotter font so the font size corresponds to ANSI standards, since the sizes of other fonts are not always consistent with ANSI standards. If you want the size of your dimensions to be the same as the font in a ***GD&T*** *label*, you must select the dimensions and change the font size or choose the plotter font.

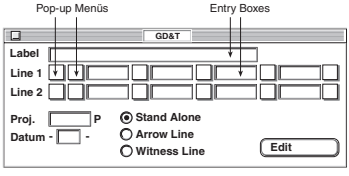
The **GD&T** dialog box resembles a ***GD&T*** *label*.



The square boxes are pop-up menus from which you can select symbols. You can use a single entry, a line, or a combination of lines as needed to specify the ***GD&T*** *dimensions* for your part.

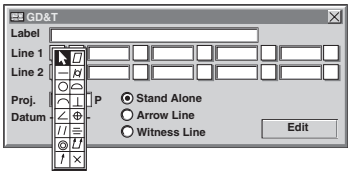
Creating a GD&T label

1. Choose **GD&T** from the **Dimension** menu.
   The **GD&T** dialog box appears.



2. Click a button (*Stand Alone*, *Arrow Line*, or *Witness Line*) to indicate the way you want the ***GD&T*** *symbol* connected to the geometry you're labelling.
3. Enter the appropriate data to create the ***GD&T*** *label*.
   Two types of entries are possible in this dialog box:

- Selecting symbols from **pop-up** menus that appear when you press the square buttons.



- Entering text in the rectangular boxes by clicking to place a text cursor, and then typing the entry.

4. Indicate the location of the label. The message line displays the instructions you need.
   - If you want a *Stand Alone* label, click the location for the upper-left corner of the ***GD&T*** *label.*
   - If you want a label connected with an arrow line or witness line, click on the geometry you wish to label, and then click on the drawing area to indicate the position for the upper-left corner of the label.

Editing a GD&T label

Once you have created a **GD&T** label, you may wish to make changes.

1. Select the **GD&T** label you wish to edit.
2. Choose **GD&T** from the **Dimension** menu.

   The **GD&T** dialog box appears, displaying the current **GD&T** information.
3. Make the changes you want in the **GD&T** dialog box.
4. Click the **Edit** button.

   The ***GD&T*** *label* changes.

Using an editing short-cut with GD&T

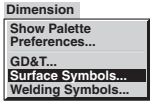
When you are editing a ***GD&T*** *label* or creating a second label, you may want to remove every entry in **Line 1** or **Line 2**.

1. Press the first box to display the **Geometric Characteristic** menu.
2. ***Drag*** to the **X** symbol.

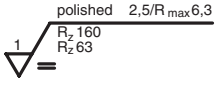
   All entries in the line disappear.

Surface Texture

The **Surface Symbols** command in the **Dimension** menu allows you to create graphical *Surface Symbols* according to *ISO 1302*.



Surface symbols describe how the surface texture of a specific part should be manufactured.



Surface Symbols

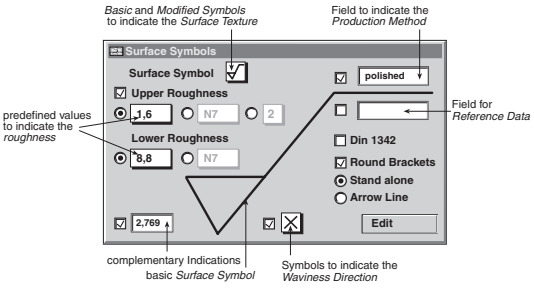
This command in the **Dimension** menu creates a graphical symbol indicating the production method for the *Surface Texture* of a part.

The **Surface Symbols** command uses *Plotter fonts* to create the symbol so the font corresponds to ***ANSI*** *standards*, since the sizes of other fonts are not always consistent with ***ANSI*** *standards*. It is not possible to change the default *Plotter* font.

It is not possible to modify the font size using the **Edit** button in the **Surface Symbol** dialog box.

The default font size is **3,5 mm**. When you want to use another font size you must modify the font size in the *Dimension Editor* before placing a *Surface Symbol*.

The **Surface Symbols** dialog box resembles a *Surface Symbol*.



Text attributes and *Arrowhead types* for *Surface Symbols* must be modified in the *dimension editor* before placing the *Surface Symbol*. It is not possible to change the default *Plotter* font.

The **Surface Symbols** dialog box contains besides several ***Text*** *entry fields* some ***pop-up*** *menus* from which you can select symbols or predefined values.

Surface Symbols
From this *pop-up menu* you can select the following *basic* and *modified basic symbols*.

Waviness Direction
From this *pop-up menu* you can select the following symbols for the waviness direction.

Roughness
From the two *pop-up menus* **Upper Roughness** and **Lower Roughness** you can select predefined values to indicate the roughness.

Creating a surface symbol

1. Choose the **Surface Symbols** command in the **Dimension** menu
   The **Surface Symbols** dialog box is displayed.
2. Enter the appropriate data to create a surface symbol by selecting symbols from the ***pop-up*** *menus* and entering data in the ***Text*** *fields*.
3. Mark an option (**Round Brackets**, **Stand Alone** or **Arrow Line**) to indicate the way you want the surface symbol connected to the geometry you're labelling.
4. Indicate the location of the surface symbol. The *Message Line* displays the instructions you need.
   For a *Stand Alone* symbol, click the location for the upper-left corner of the symbol.
   If you want the symbol connected with an *Arrow Line*, click on the geometry you want to label, and then indicate on the drawing area the position for the upper-left corner of the symbol with a single mouse click.

Some ***pop-up*** *menus* have an *option button* in front of the menu. You have to mark this button first before you can select any value from this ***pop-up*** *menu*.

Editing a surface symbol

Once you have created a *Surface Symbol*, you may wish to make changes.

1. Select the *Surface Symbol* you wish to edit.
2. Choose the **Surface Symbols** command in the **Dimension** menu
   The **Surface Symbols** dialog box is displayed.
3. Make the changes you want in the **Surface Symbol** dialog box.
4. Click the **Edit** button.
   The *Surface Symbol* changes accordingly.

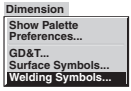
It is not possible to modify the *Font Size* or *Font Style* using the **Edit** button in the **Surface Symbol** dialog box. It is not possible to change the default *Plotter* font. The *Font Size* can be modified in the *dimension editor* but will affect only newly created *Surface Symbols*.

Closing the Surface Symbols dialog box

The **Surface Symbol** dialog box is closed automatically when you select a function from the *Function palette*.

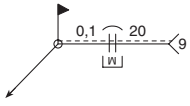
Symbolic Representation of Welding Joints

The **Welding Symbols** command in the **Dimension** menu allows you to create *Welding Symbols* according to *DIN 1912* (contains *ISO 2553*).



The graphical symbol prescribes the method how welded joints should be manufactured. The graphical representation contains an elementary symbol that can be completed by the following informations:

- combined and supplementary symbols,
- welds dimensions,
- complementary indications.



Welding Symbols

This command in the **Dimension** menu creates a symbol for the graphical representation of welded joints.

The **Welding Symbols** command uses *Plotter* fonts to create the symbol so the font corresponds to ***ANSI*** *standards*, since the sizes of other fonts are not always consistent with ***ANSI*** *standards*. It is not possible to change the default *Plotter* font.

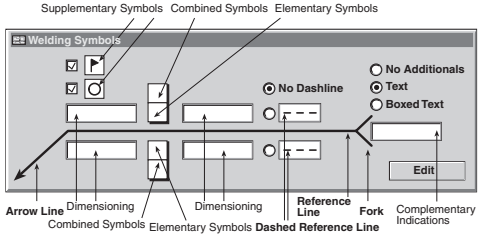
It is not possible to modify the font size using the **Edit** button in the **Welding Symbol** dialog box.

The default font size is **3,5 mm**. When you want to use another font size you must modify the font size in the *dimension editor* before placing the *Welding Symbol*.

The **Welding Symbols** dialog box resembles a *Welding Symbol*.

8

Text attributes and *Arrowhead types* for *Welding Symbols* must be modified in the *dimension editor* before placing the *Welding Symbol*. It is not possible to change the default *Plotter* font.



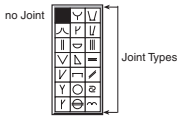
The **Welding Symbols** dialog box contains besides several ***Text*** entry fields some ***pop-up*** *menus* from which you can select symbols or predefined values.

Basic Symbol — The displayed basic *Welding Symbol* contains an *Arrow Line*, a *Reference Line*, an optional *Upper* and *Lower Dashed Reference Line* and the *forked end* of the *Reference Line* for complementary indications.

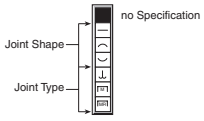
Dashed Reference Line — The *dashed Reference Line* indicates whether the joint should be welded from the *Reference Side* (Dashed Line above the Reference Line) or from the opposite side (Dashed Line below the Reference Line).

You can choose from the options *No Dash Line*, *Upper Dash Line* and *Lower Dash Line* by clicking on the related option button.

Elementary Symbols — *Elementary Symbols* indicate the joint type.



Combined Symbols — Elementary symbols can be supplemented by a *Combined Symbol* indicating the *Shape* and *Production method* of the joint.



Dimensions Each *Welding Symbol* can contain dimension data. Dimensions for the *joint thickness* are entered on the left side of the *Elementary Symbol*, dimensions for the *joint length* on the right side.

Supplementary Symbols There exist two types of symbols: *Circular Joints* (indicated by a *circle*) and *Longitudinal Joints* (indicated by a *flag*). Both types may be combined in a symbol.

Complementary Indications There are three complementary indications possible:

No additional The *Reference Line* of the welding symbol is created **without** a fork at the end.

Only Text This option is used when all complementary indications are placed in the related ***Text*** *entry field*. The *Reference Line* of the welding symbol is created **with** a fork at the end.

Boxed Text This option is used when all complementary indications are listed separately and the ***Text*** *entry field* contains only corresponding reference data. The *Reference Line* of the welding symbol is created **with** a closed forked at the end.

Creating a welding symbol

1. Choose the **Welding Symbols** command in the **Dimension** menu
   The **Welding Symbols** dialog box is displayed.
2. Enter the appropriate data to create the welding symbol by selecting symbols from the ***pop-up*** *menus* and entering data in the ***Text*** *fields*.
3. Mark the option *No additional* if no complementary indications are required or one of the options *Only Text* or *Boxed Text* if complementary indications are required.
4. Indicate the location of the welding symbol. The *Message Line* displays the instructions you need.

Some ***pop-up*** *menus* have an *option button* in front of the menu. You have to mark this button first before you can select any value from this ***pop-up*** *menu*.

Editing a welding symbol

Once you have created a *Welding Symbol*, you may wish to make chances.

1. Select the *Welding Symbol* you wish to edit.
2. Choose the **Welding Symbols** command in the **Dimension** menu
   The **Welding Symbols** dialog box is displayed.
3. Make the changes you want in the **Welding Symbol** dialog box.
4. Click the **Edit** button.
   The *Welding Symbol* changes accordingly.

It is not possible to modify the *Font Size* or *Font Style* using the **Edit** button in the **Welding Symbol** dialog box. It is not possible to change the default *Plotter* font. The *Font Size* can be modified in the *dimension editor* but will affect only newly created *Welding Symbols*.

Closing the Welding Symbols dialog box

The **Welding Symbol** dialog box is closed automatically when you select a function from the *Function palette*.

CHAPTER

9

Basic Drafting

Zooming

Screen Redraws

The Drafting Process

Drawing At Full Scale

Paper Size and Drawing Scale

Scaling Drawings

Scaling Commands

Printing or Plotting a Drawing

Basic Drafting

This chapter describes techniques for *Printing* and *Scaling*. There are two different ways to change the drawing size:

- *Zooming*
  You can change the magnification of your drawing by zooming in and out using commands, zoom tools, and strokes. Zooming changes the visual size of the drawing, never the physical size of the objects.
- *Scaling*
  Scaling means to set the visual size of the drawing to fit to the selected paper size when printing or plotting. Scaling too changes the visual size of the drawing, never the physical size of the objects.

This chapter describes the following techniques:

- Zooming
- Full-scale drawing
- How scaling affects *text*, *dimensions*, *line patterns* and *hatch patterns*
- Scaling commands
- Printing or plotting drawings on standard paper sizes

Most drafters begin a project by deciding how to scale the part first. Using ***DraftBoard*** you create all parts at full scale (**1:1**).

You never physically scale the part, instead you set the visual scale of the geometry (the visual size of the drawing) to fit to the selected paper size when printing or plotting the drawing.

Zooming

DraftBoard gives you several ways to *zoom in* or *zoom out* on your drawing.

Zoom Commands

You find four *Zoom commands* in the **Arrange** menu: **Zoom In**, **Zoom Out**, **Zoom All** and **Zoom Previous**. Choosing the **Zoom Previous** command takes you to the last magnification. If you want to zoom a particular area, use the **Stroke** feature or the *Magnifying* tool for zooming. *Strokes* are described later in this chapter.

*Important: Each Zooming is followed by a screen redraw. You can press the **Pause** or **Esc** key (Macintosh: only **Esc** key) to stop redrawing the screen.*

Zoom All **Ctrl+F** [Macintosh: ⌘F]
This command in the **Arrange** menu zooms in or out to make all parts of the drawing visible on the screen.

Zoom In **Ctrl+]** Macintosh: ⌘]
This command in the **Arrange** menu zooms in on the center of the screen by a factor of two. You cannot specify a particular area for enlargement.

Zoom Out **Ctrl+[** Macintosh: ⌘[
This command in the **Arrange** menu zooms out from the center of the screen by a factor of two. You cannot specify the area of reduction.

Zoom Previous
This command in the **Arrange** menu zooms to the previous magnification, as many as five times.

Zoom Tools

Select the appropriate zoom tool from the **View Control** tool palette.

*Important: Each Zooming is followed by a screen redraw. You can press the **Pause** or **Esc** key (Macintosh: only **Esc** key) to stop redrawing the screen.*

Using the magnifying glass tool

With the **Zoom** tools you ***drag*** a box around an area on the screen, so only that area is magnified on the screen.

1. Select a **Zoom Tool** from the tool palette.
2. With the left mouse button pressed ***drag*** a box around an area on the screen you want to magnify or reduce.
3. Release the mouse button.
   The content of the ***dragged*** box is made visible on the screen.

Zoom In Tool

This tool zooms in by the specified factor. The default factor is **2**. This is a visual rather than a physical change.

To zoom into or out of a drawing you can use as well the mouse wheel, if your mouse is supporting this function.

When you click in the drawing area, that position is redisplayed in the center of the screen, and the drawing is enlarged by a factor of **2**.

You can also ***drag*** a box around an area, so only that area is magnified.

The status line shows the current zoom scale. If you enter a different scale, that scale takes effect when you press **Enter**. Pressing the **Ctrl** key (*Macintosh:* ***Option*** *key*) while using this tool causes a **Zoom Out**.

Zoom Out Tool

This tool zooms out by the specified factor. The default factor is **0.5.** This is a visual rather than a physical change.

When you click in the drawing area, that position is displayed in the center of the screen and the drawing is reduced by a factor of **0.5**.

The status line shows the current zoom scale. If you enter a different scale, that scale takes effect when you press **Enter**.

Pressing the **Ctrl** key (*Macintosh:* ***Option*** *key*) while using this tool causes a **Zoom In**.

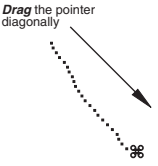
Zoom Stroke

You can use stroke commands to zoom—magnify or reduce your drawing. *Stroke commands* are useful because they don't require you to get out of the tool you're using to zoom. Hold down the **Ctrl** and **Shift** key (***Macintosh:*** ⌘ key) and ***drag*** the pointer that appears diagonally across the screen as described below. The pointer takes on the shape shown below when you hold down the **Ctrl** and **Shift** key (*Macintosh:* ⌘ key).

Using Stroke zoom

Note: *For* ***Zoom In*** *and* ***Zoom Out****, the size and location of the stroke rectangle is important for determining the result of the* ***Zoom*** *operation. For* ***Zoom Previous****, the size and location of the stroke rectangle is irrelevant. All cases just give the previous magnification.*

Depending of the stroke direction you can execute different *Stroke commands*:



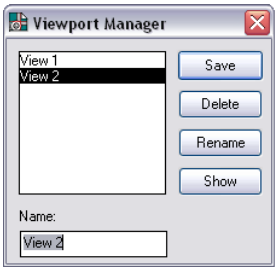
Drag **Diagonally**	**Result**
Upper left to lower right	Zoom-in enlargement centered over the stroked area.
Lower right to upper left	Reverses **Zoom In** stroke to the previous magnification.
Upper right to lower left	Zoom-out reduction-the current screen reduces to the size of the area defined by the stroke.
Lower left to upper right	Reverses **Zoom Out** stroke to previous magnification.

Saving Drawing Areas

The **Viewport Manager** command in the **Arrange** menu allows you to save screen contents with their current *Zoom factor*. The **Viewport Manager** offers a big help when navigating through large 2D drawings.

Viewport Manager

When you select this command the following dialog box is displayed:



The **Viewport Manager** dialog box contains the following elements:

New When you click this button the current content of the screen is saved with the name **View 1,2,3...** and is displayed in the **View** list. You can rename the default name **View 1** using the **Rename** button.

Existing views cannot be renamed.

Delete This button allows you to delete the view selected in the *view list*.

Rename This button allows you to rename a view selected in the view list with the name entered into the **Rename** field.

Name Into this field you can enter a name for a view selected in the *view list* and rename it with the **Rename** button.

Show This button displays the saved drawing area of the selected view on the screen.

Saving a View

1. Select the **View Manager** command in the **Arrange** menu.
   The **Viewport Manager** dialog box is displayed.
2. Zoom into the desired drawing area using the **Zoom In** or **Zoom Out** tool in the **Views** subpalette.
3. Click the **Save** button.
   A new view with the default name **View 1** is created and the displayed screen content is saved.
4. Enter a proper name for the new view into the **Rename** field.
5. Click the **Rename** button.
   The selected view is a renamed with the name entered into the **Rename** field.

Renaming a View

1. Select the **View Manager** command in the **Arrange** menu.
   The **Viewport Manager** dialog box is displayed.
2. Select the view you want to rename in the *View list*.
3. Enter a new name for the view selected in the *view list*.
4. Click the **Rename** button.
   The view selected in the view list is a renamed with the name entered into the **Rename** field.

Showing Views

1. Select the **View Manager** command in the **Arrange** menu.
   The **Viewport Manager** dialog box is displayed.
2. Select the view you want to rename in the *View list*.
3. Click the **Show** button.
   The screen content saved with this view is displayed on the screen. You can display a view as well by double-clicking the view name in the view list.

Deleting Views

1. Select the **View Manager** command in the **Arrange** menu.
   The **Viewport Manager** dialog box is displayed.
2. Select the view you want to rename in the *View list*.
3. Click the **Delete** button.
   The selected view is deleted.

Panning Screen Contents

The tool **Pan Drawing area** allows to move the content of a screen. You find this tool in the **Views** subpalette.

Pan Drawing area

This tool displays a hand icon that allows you to move the screen area while the mouse button is pressed.

The status line contains no boxes.

Panning a drawing area

1. Select the **Pan drawing area** tool in the **Views** subpalette of the *Tool Palette*.
2. Press the mouse button and ***move*** the content of the screen while the mouse button is pressed.

Screen Appearance

DraftBoard offers several commands to control screen appearance.

Redraw Screen Ctrl+R [Macintosh: ⌘R]

This command in the **Arrange** menu refreshes the screen. When you make changes to your constructions, the geometry may not be redrawn cleanly in the drawing area.

Redrawing the screen

To redraw all of the geometry and remove extraneous geometry and points, choose the **Redraw Screen** command from the **Arrange** menu.

Stopping a screen refresh

You can press the **Esc** or **Break** key **(**Macintosh: ⌘key) to stop the redrawing of the screen.

Important*: Under Windows you should use the* ***Break*** *key instead of the* ***Esc*** *key, since Windows displays the Task manager when the* ***Redraw screen*** *command was initiated by the key combination* ***Ctrl+R****.*

Invert

This command in the **Preferences** submenu in the **Layout** menu allows you to switch the color of the drawing area from *White* to *Black* and vice versa. The default color is *White*.

The Drafting Process

When creating a drawing you have to consider several items to guarantee that the drawing is printed within a *standard drawing format* according to the *standard drawing rules*. While you don't have to follow the these suggestions in order, you may want to use the following scheme until you have enough experience to know how each step affects the overall project.

Creating a drawing

1. Start ***DraftBoard***.
   DraftBoard automatically displays an **Untitled** document.
2. If necessary, set the **Units** of measure you want to use with the **Preferences** command in the **Layout** menu.
3. Use the **Drawing Size** command in the **Layout** menu to set the drafting scale to **1:1**.

The precision you specify for the selected unit defines only the number of digits you can enter in the status line, not the precision of the object geometry, that is always saved with double precision. Therefore you can change the precision at any time during the drafting process.

4. Construct the geometry at full size, saving the construction as you work. Use the **Zoom** commands and tools to display the part at an appropriate magnification so that you can see what you are doing.
5. Use the **Print Setup** command (*Macintosh:* ***Page Setup***) in the **File** menu to set the paper size you will use for plotting or printing.
6. Use the **Drawing Size** command in the **Layout** menu to set the visual scale of your drawing to fit on the specified paper size for plotting or printing.

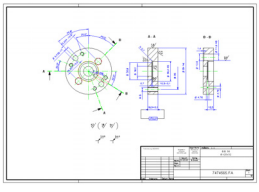
   You may specify the drafting scale, such as **1:100** or **1cm:1m** for **1** centimeter to represent **1** meter.

   If you are not concerned about scale and you only want the final geometry to fit on the selected paper size, you can specify **Fit** to adjust the geometry automatically to the paper size.
7. *Dimension* the part and *add text*.
8. Import a *standard drawing format* with the **Import** command in the **File** menu from the **Layout** folder.
9. With the **Preview Layout** command in the **File** menu you can check size and orientation of the printing area that is displayed by a grey border to make final adjustment if necessary.
10. Choose **Print** from the **File** menu.

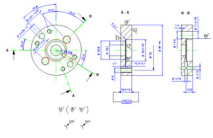
    The drawing is printed or plotted on the specified printer or plotter. *Text* and *dimensions* are printed in the size you specified, since they were added **after** you scaled the drawing.

    When you have set for example the size for *text* and *dimensions* to **3,5 mm**, *text* and *dimensions* will be printed in a size of **3,5 mm**.

How you have to proceed if you added *text* and *dimensions* **before** you scaled the drawing is described in the following sections of this chapter.



Flange plotted with a standard drawing format including block and title.



Flange plotted without a standard drawing format.

The above described procedure recommends to scale the drawing (to adjust the visual scale to the specified paper size) **before** you add *text* and *dimensions*.

Of course you can scale drawings in ***DraftBoard*** after you have added *text* and *dimensions*. But that requires that you understand the different techniques ***DraftBoard*** uses for scaling. These procedures are described in the following sections of this chapter.

Drawing at Full Scale

In ***DraftBoard*** you create all parts at its actual size. It doesn't matter whether you design a large industry building or a small detailed blueprint.

When printing you adjust the visual scale (the view of the drawing) to fit the paper you will use. The true size of all objects saved in the ***DraftBoard*** *Database* is never affected by scaling.

DraftBoard saves all objects at their true measurements, regardless of

- which drawing scale you specify with the **Drawing Size** or the **Sheet Into View** command
- what extend you magnified the drawing with the *Zoom commands*.

With these commands you only change the visual size of the objects on the screen or on the paper but never its true size saved in the ***DraftBoard*** *database*.

In this way the part dimensions to its true-to-life measurements regardless how they are displayed on the screen or printed on the paper.

Drawing at full scale has the following advantages:

- scaling mistakes are eliminated
- dimensions are automatic (you must dimension manually if you do not draw at full scale)
- associative dimensions update when the object is edited (manual dimensions do not)
- the size relationship of imported parts is compatible.
- calculations for *2D analysis* are accurate

Creating and editing objects at full size

When you open a new ***DraftBoard*** document, the drawing area is a sheet that is infinitely large so that you can design anything at full size. For a simplistic example, here's how to draw and view a line *24 meters* long:

1. Draw a line.
2. For the Length of the line type **24 m** in the **Length** field of the status line.

   The line extends off the screen.

Zoom All magnifies or reduces all objects on your drawing to fill the screen—regardless of how big or small the objects are.

3. Choose **Zoom All** from the **Arrange** menu.

   The entire **24** meter line is visible on the screen.

While zooming ***DraftBoard*** calculates the scale required to display the line at its full length on the screen. When you zoom again ***DraftBoard*** doesn't use the scaled length to calculate the new scale (this would lead to scaling mistakes the more often you scale) but its original true length saved in the ***DraftBoard*** *database*. The *Zooming commands* allow you to create accurate full-scale drawings and display them at the magnification required.

Paper Size and Drawing Scale

Before you can check if your drawing can be printed at full size on a specific paper format you have to specify a paper size and orientation for your installed printer or plotter.

Specifying the paper size

To specify a paper size you choose the **Print setup** command (*Macintosh:* ***Page setup***)

WIN **Print Setup** WIN

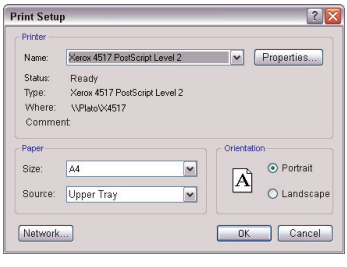
This command in the **File** menu allows you to select a printer, set the page size, orientation, and other options.

Setting up the page WIN

1. Select **Print Setup** in the **File** menu.

   The following the dialog box is displayed.

In case your printer is not listed in the list of available printers you must first install your printer under Windows. A detailed description how to install a printer under *Windows* can be found in the *Windows Reference Manual*.



2. Choose your printer in the **Specific Printer** field if it is not installed as the **Default Printer**.
3. Select the paper size from the **Paper Size** *pop-up* menu.
4. Select the paper source from the **Paper Source** *pop-up* menu
5. Specify the **Orientation** (*Portrait* or *Landscape*).

6. Click the **Option** button.

   Specify in the displayed dialog box if you want to print your drawing to a printer or save it to a file.

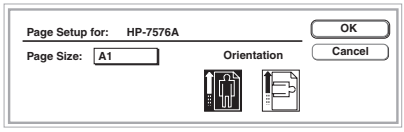
7. Click the **OK** button.

Page Setup MAC

This command in the **File** menu allows you to select a printer, set the page size, orientation, and other options.

Setting up the page MAC MAC

1. Select the plotter or printer you will use. Contact *Technical Support* if you need help setting up a plotter.



2. Select the **Page Size** from the **Page Size** pop-up menu.
3. Specify the **Orientation**.
4. Click the **OK** button.

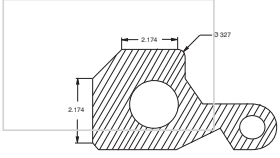
Checking the View Scale

After you have specified the *paper format* you must check if the objects of your drawing are too large or too small in relation to the selected paper format and if its view has to be scaled to fit on the specified paper size.

The **Preview Layout** command in the **File** menu allows you to check the size ratio between the objects and the specified paper size.

Preview Layout

This command in the **File** menu displays the *printing area* of the selected paper at its true size and selected orientation. If you have not specified a *printing/plotting size*, the default size is used.



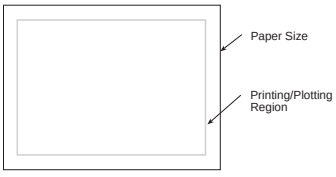
If the drawing does not fit into the displayed rectangle representing the printing area of the selected paper size, you must scale the drawing.

Printing/Plotting region

The *printing/plotting region* is displayed when choosing the **Preview Layout** command and the **Drawing size** command.

With both commands a grey rectangle appears in the drawing area. This rectangle represents the *printing/plotting region* of the page so you can see how your drawing fits the paper size.

The *plotting region* is smaller than the actual *page size* because most printers and plotters cannot plot to the edge of the paper. If you want to see the plottable outline all the time, click the **Always Display** button.



DraftBoard shows always the printable area

When the boundary of the printing/plotting region appears, you can ***drag*** the boundary rectangle around the drawing area.

When you choose the **Preview Layout** command again or select any tool from the tool palette the *Printing/plotting region* disappears.

If the grey boundary representing the page size is not visible after scaling choose **Zoom all** while the **Drawing Size** dialog box is displayed.

The *printing/plotting region* is displayed filling the screen with the **Origin** (**0,0**) placed in the center of the page. If you cannot see your drawing on the screen you select the **Zoom All** command in the **Arrange** menu.

By the size relationship between the existing object geometry and the displayed *printing/plotting region* you can check visually if you have to scale the view of your drawing. To do so you have to know how the different elements of a drawing such as *geometry*, *text*, *line patterns* behave while scaling, this described in the next section.

Scaling Drawings

Whenever the created object geometry is too large or too small in relation to the specified paper format drawings must be scaled.

Scaling affects elements of a drawing differently. While scaling ***DraftBoard*** distinguishes between the following elements:

- object geometry
- hatch patterns
- line patterns
- text
- dimensions

For scaling you can use the **Drawing Size** command in the **Layout** menu or the **Sheet Into View** command in the **Views** menu. Since both commands affect elements of a drawing the same way while scaling the following sections describe only scaling with the **Drawing Size** command.

Scaling Object geometry

If the grey boundary representing the page size is not visible after scaling choose **Zoom all** while the **Drawing Size** dialog box is displayed..

Let's assume that a drawing exists only of object geometry created with solid lines.

In that case you can see the result of the scaling only by the changed size of the paper format represented by a grey rectangle in the drawing area for the *printing/plotting region* of the page.

If the grey rectangle for the page size isn't visible you can't see the reduced or increased size of the object geometry caused by the scaling command, since the size of the individual objects didn't change in relation to each other but only in relation to the paper size.

Scaling Line patterns and Crosshatch patterns

In case the *dashed* or *dotted line patterns* are not displayed correctly because the scale factor is too large or too small you can adjust the line pattern using **Edit Pattern** the command in the **Pen** menu.

Remember: Lines with *dashed* or *dotted line patterns* and *Hatch patterns* are never scalded in ***DraftBoard*** but always displayed at their true size.

When you use *hatches* and *dashed lines* in a drawing, the line dashes and hatch patterns are displayed at their true size.

When these *line dashes* are very small in relation to the remaining object geometry (size of *object geometry* in meters and size of *line dashes* in millimeters) you can't identify these *line dashes* on the screen before scaling, since they are displayed at actual size.

When you scale the drawing the object geometry is scaled while the dashes of the lines and hatch patterns keep their true size. Therefore *line and hatch patterns* change size in relation to each other.

When you reduce the view scale of the drawing the size of the object geometry is reduced in relation to the dashed lines with the result that *line* and *hatch patterns* are now visible on the screen and on a print out.

Scaling Text and Dimensions

If your unscaled construction is very large or very small compared to the size of added *text* and *dimensions* you have to scale the view of the drawing.

While scaling ***DraftBoard*** distinguishes if the options

- **Keep Text size**
- **Keep Dimension Text size**

are checked or not. When both options are checked *text* and *dimensions* are **not** scaled and keep the size you have assigned in the **Text** menu or in the **Dimension Editor**. With these options checked the size of the object geometry will be increased or reduced in relation to *text* and *dimensions* .

However when you scale a drawing without having these options checked

- *text* and *dimensions* are scaled

- *text* and *dimensions* that you add **after** scaling are not scaled and displayed at true size.

This feature allows you to scale *text* and *dimensions* if necessary.

Options Keep Text size and Keep Dimension Text size

The options **Keep Text size** and **Keep Dimension Text** size are only available with the **Drawing Size** command not with the **Sheet Into View** command.

Therefore you must when scaling a drawing using the **Sheet Into View** command, select after scaling all text and dimensions and assign again the *original text size* in the **Text** menu or the *Dimension Editor*.

It is advisable to use the **Selection mask** to select all text and dimensions in case you have to assign the accurate text size **after** scaling.

This procedure is necessary since ***DraftBoard*** scales *existing* text and dimensions with the same scale factor as the object geometry and keeps *text size* and *dimension text size* only for future text and dimensions which are added after scaling.

Text and *dimensions* you place **after** scaling will be displayed at true size even if the options *Keep Text Size* and *Dimension Text Size* are active or scaling was executed with the **Sheet Into View** command.

Plotter Fonts

When you are using a plotter, and the text should be displayed at its accurate size, you should specify a plotter font for text and dimensions on your drawing, since all other fonts are proportional and may differ in their appearance slightly from the assigned size.

All *Plotter Fonts* available with ***DraftBoard*** are listed in *Appendix **A*** of this *Reference Guide*.

You can also specify different text styles (such as *italic* or *bold)* for plotter fonts in the **Text** menu and generate special characters and accents as described in the *Appendix* of this manual.

Scaling Rules

While scaling the different elements of a drawing behave as follows:

- *object geometry* is **always** scaled
- *line* and *hatch patterns* are **never** scaled
- *text* and *dimensions* are **not** scaled if the options *keep text size* and *keep dimension text size* are active.
- *text* and *dimensions* are **always** scaled if the options *keep text size* and *keep dimension text size* are not checked.

To keep *text* and *dimensions* at a constant size while scaling and having the options *keep text size* and *keep dimension text size* not checked you have to:

It is advisable to use the **Selection mask** to select all text and dimensions in case you have to assign the accurate text size **after** scaling.

- add *text* and *dimensions* **after** scaling

  or
- you must select all *text* and *dimensions* and assign their text size again.

Scaling Commands

If your unscaled construction is very large or very small compared to the size of the specified page format you have to adjust the view scale of your drawing to fit the selected paper size. To do this you can use

- the **Drawing Size** command in the **Layout** menu

  or
- the **Sheet Into View** command in the **Views** menu.

The **Drawing Size** command with its options *Keep Text Size* and *Keep Dimensions Text Size* offers the better control for text and dimensions when scaling.

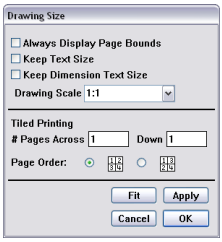
Whereas the **Sheet Into View** command imports automatically a standard drawing format and scales a drawing at the same time, but doesn't keep the text size when scaling.

Drawing Size

This command in the **Layout** menu allows you to see the size of the maximum *plotting/printing* area relative to the drawing and to specify the scaling of the drawing so it fits the paper size and orientation set by the **Print Setup** command (*Macintosh:* ***Page Setup***).

When you choose the **Drawing Size** command the following dialog box is displayed:

Before you choose the **Drawing Size** command in the **Layout** menu, you should use *zoom out* until your drawing fills half the screen size.



The **Drawing Size** dialog box allows the following settings:

Always display Page Bounds — When you check this option the rectangle for the *printing/plotting region* is always displayed at its true size in the drawing area even if the **Drawing Size** dialog box is closed.

Keep Text Size — When this option is checked *text* is **not** scaled but keeps the *text size* set before scaling.

Keep Dimension Text Size — When this option is checked *dimensions* are **not** scaled but keep the *text size* set before scaling under **Preferences** in the **Dimension** menu.

Drawing Scale — In the **Scale** box you enter a *scale factor* for your drawing specifying the ratio between the *printed* and the *true size* of your drawing.

To specify a scale you can press the arrow beside the **Scale** box and ***drag*** to one of the standard ratios from the menu or enter real values, such as **1cm: 1m**, which is converted to **1:100** by ***DraftBoard***.

Fit — When you click this button the drawing is scaled to fit the printing region of the specified paper size. The calculated scale factor is displayed in the **Scale** box and the **Drawing Size** dialog box stays open.

Apply — When you click the **Apply** button the drawing is scaled according to the selected factor and you can see the specified scaling on the drawing area. The **Drawing Size** dialog box stays open.

OK — When you click the **OK** button the drawing is scaled according to the specified scale factor. The **Drawing Size** dialog box is closed and the rectangle of the printing region disappears on the screen when the option **always display Page Bounds** is not checked.

Setting up the drawing size using the Drawing Size command

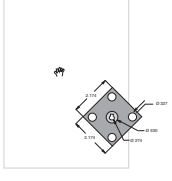
The **Cancel** Button on the **Drawing Size** dialog box doesn't abort changes if a **Fit** or **Apply** are applied first.

1. Choose **Drawing Size** from the **Layout** menu.

   The **Drawing Size** dialog box is displayed.

   With the dialog the grey rectangle for the *printing/plotting region* is displayed at its true size on the drawing area. When the boundary of the *printing/plotting region* appears, you can ***drag*** the boundary rectangle around the drawing area.

If you want to change the *Page orientation* of the displayed page boundary you have to use the **Print Setup** Command (*Macintosh:* ***Page Setup***).



2. Specify the scale ratio.

   You can press the arrow beside the **Scale** box and ***drag*** to one of the standard ratios from the menu or double-click the **Scale** box and enter a scale (the ratio) you want to use. You can enter real values, such as **1cm:1m**, which is converted to **1:100** by ***DraftBoard***.

If you don't need a specific scale, you can click the **Fit** button to scale the drawing to fit the paper.

3. When *text* and *dimensions* should keep their true size choose the options **Keep Text Size** and **Keep Dimension Text Size**. If you don't select these options you have to adjust *text* and *dimensions* **after** scaling.
4. When you want to control the scaling on the screen click the **Apply** button.

   The drawing is scaled and the **Drawing Size** dialog box remains open. The geometry itself does not change scale; it only changes visually, not physically.
5. If you want to print multiple sheets and tape them together, follow the directions for *Tiled Printing* described later in this chapter.
6. Click **OK**.

Setting up the drawing size using the Sheet Into View command

For scaling a drawing you can use as well the **Sheet Into View** command in the **Views** menu. The **Sheet Into View** command imports automatically a standard drawing format and scales a drawing at the same time to fit into a view window placed within the drawing format. The **Sheet Into View** command does not keep the text size when scaling.

You find a detailed description of the **Sheet Into View** command in the *Chapter **10** Structuring Drawings.*

1. Create the object geometry.
2. Choose **Zoom All** from the **Arrange** menu.

   The screen is filled with the object geometry.
3. Choose the **Sheet Into View** command from the **Views** menu.

   The **Sheet Into View** dialog box is displayed.
4. Enter a *Scale factor* for the drawing into the **Scale** box. This *Scale factor* should correspond with the selected paper size.
5. Click **OK**.

   The whole object geometry is scaled and displayed in an independent view window.
6. Add *Text* and *Dimensions*.

   Text and dimensions are placed unscaled at true size.

   Since the options *Keep Text Size* and *Keep Dimension Text Size* are not available with the **Sheet Into View** command you must select all *text* and *dimensions* and assign their text size again.

It is advisable to use the **Selection mask** to select all text and dimensions in case you have to assign the accurate text size **after** scaling.

Printing or Plotting a Drawing

DraftBoard prints and plots on most printers and plotters supported by your computer. After you have followed the manufacturer's instructions for installing and setting up the printer or plotter, you can size your drawing for the paper you will be using.

Print Ctrl+P [Macintosh: P]

This command in the **File** menu prints or plots the current document as specified in **Print Setup** (*Macintosh:* ***Page Setup***).

The **Print Setup** command (*Macintosh:* ***Page Setup***) allows to change the current plotter or printer and to set options such as *Paper Size* and *Paper Orientation.*

The area printed or plotted is the portion that fits on the page size when the origin (**0,0**) is placed in the center of the page. Use the **Drawing Size** command from the **Layout** menu to reposition the *print/plot region*.

By repositioning the *printing/plotting region* it is possible to print only part of a drawing, since the only part that is printed is the part that lies within the printing region.

Plotting to a file

You can plot to a file rather than to a plotter or printer. In that way, you don't have to have a plotter attached to your computer, you can have someone else plot the drawing without having a copy of ***DraftBoard*** on the plotter's computer.

If you want to use ***DraftBoard*** *drawings* in a *Text Editor* or *Layout program* the ***EPS*** *file format* is recommended, since it offers the best quality and allows to scale the graphic after placing without any quality loss. How to create an ***EPS*** *file* is described in *Chapter **11** DraftBoard Documents.*

The type of plotter you choose when you are setting up the page determines the format of the plot file. For example, if you choose a **PostScript** printer, the file format will be **E**ncapsulated **P**ost**S**cript;

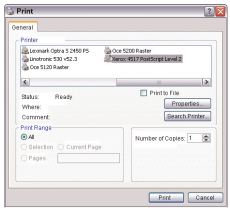
and the **HPGL** language is used when you select **Hewlett Packard** plotters. The computer that finally plots the file must have an application compatible with the file format of your printer or plotter.

WIN

Plotting to a file under Windows

1. Choose **Print** command in the **File** menu.

   The **Print** dialog box is displayed.



2. Mark the **Print to file** option.
3. Click **OK**.

   The **Print to file** dialog box is displayed.
4. Enter a name for the print file.
5. Click **OK**.

   The drawing will be printed to file and saved under the name you specified.

Mac

Plotting to a file on a Macintosh computer

1. Choose **Chooser** from the **Apple** menu.
2. Select a printer or plotter from the **Chooser** list.

   A message appears telling you to confirm settings in **Page Setup** in the **File** menu.
3. Click **Continue**.
4. Click **Setup** in the **Chooser** dialog box.
5. Specify the printer and options you want to use.
6. Choose **Print** from the **File** menu.
7. Click the **File** button in the lower-right corner of the **Print** dialog box.
8. Enter the *file name* and click **OK**.

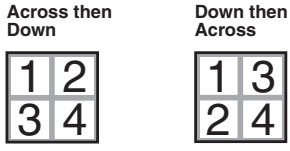
   The drawing will be printed to file and saved under the name you specified.

9

Tiled Printing

If you are using a printer rather than a plotter, you may want to tape pages together to get a larger drawing than laser printers allow. Specify the layout for those pages by choosing **Drawing Size** from the **Layout** menu.

1. Choose **Drawing Size** from the **Layout** menu.
2. Specify the number of pages *across* for the final drawing.
3. Specify the number of pages *down* for the final drawing.
4. Specify the order for printing the pages (*across then down* or *down then across*).



5. Click the **Fit** button.

   The drawing will be scaled to fit to the number of pages you specified.
6. With the mouse button pressed reposition the displayed rectangles representing the number of pages you specified to cover these objects you want to print.
7. Click **OK**.

   Your drawing is scaled to fit the number of pages and layout, as you have specified.

CHAPTER

10

Structuring Drawings

Layer

Sheets

Models

Associative Views

Drawing Formats

Working with Sheets, Views and Models

Structuring Drawings

DraftBoard provides the following features to structure drawings:

- Layers
- Sheets
- Associative Detail Views
- Models

These tools allow you to use ***DraftBoard*** individually for your design tasks. Especially the combination of *Sheets*, *Views* and *Models* add a great amount of flexibility to the design process.

Layers

Layers are particularly useful in helping you view and plot complex drawings. For example, when you dimension a part, the dimensions can be placed on a separate layer which you can display or not, as your needs require. *Layers* allow you to plot different versions of the same document for quick and easy specialized blueprints.

Layers are often described as transparent films. But actual they are just another method to group objects.

- You can hide the *dimension layer* to exhibit the idea of a design to a planning team and show the dimensions when presenting the drawing to engineers.
- You can hide some drawing components when you plot. For example, you can hide the construction layer so that construction lines and geometry don't plot, but they remain in the drawing ready for use when you want to alter it.
- You can construct different layouts using one layer as the basis. For example, you can use a floor plan and then construct the electrical and plumbing plans on different layers. In that way, you can turn off one layer (the electrical plan, for instance) and print the other (the plumbing plan) with the floor plan.

DraftBoard allows you to have up to **256** visible or hidden layers in your drawing. Layer names are limited to **79** characters.

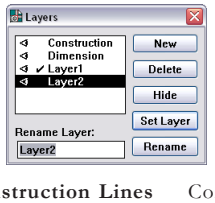
Since the ***DXF*** *format* supports only **31** characters you should limit layer names to **31** characters to guarantee a smooth ***DXF*** *Export.*

Layers Ctrl+L [Macintosh: ⌘L]

This command in the **Layout** menu allows you to *create*, *delete*, and *rename* layers, as well as set layer specifications. The *work layer* is the active layer—the layer on which the current construction is created.

Note: *You cannot hide or delete a layer that is the current (active) work layer.*

The visible layers are indicated in the list box by a **checkmark** and a **dot** indicates the work layer.



DraftBoard includes a **Construction layer**, a **Dimension layer** and a **TitleBlocks** layer by default with each drawing.

Construction Lines Construction lines automatically go on the construction layer.

Dimensions If you create a dimension layer, all dimensions added after you create the layer go on the dimension layer.

Note: *The* ***Construction*** *layer will automatically accept all* ***Construction lines*** *created with stroke commands or with the* ***Construction*** *dialog box. All lines of "construction" line type created with the* ***Single Line*** *tool will go onto the current work layer.*

You can edit any geometry or text that is visible, regardless of which layer it is on. If you want to make some geometry unselectable but still visible, use the **Selection Mask** command in the **Edit** menu.

You can use layers for complex drawings so you can hide some objects to simplify the drawing visually or print only specific parts of the drawing. You can think of visible layers as transparent pages and hidden layers as invisible pages. Although you cannot see objects on hidden layers, they do exist. The layer must be visible for you to select objects on the layer. For example, if there are objects on a hidden layer and you choose **Select All**, then delete all objects, those objects on the hidden layer are not deleted.

You can also specify the selectability of layers with the **Selection Mask** command in the **Edit** menu.

Creating a new layer

You can also type a name and click **New**.

Click **New** to create another layer. You can name the layer by typing the name in the **Rename** entry box and clicking **Rename**. You can have as many as **256** layers.

Renaming a layer

Layer names are limited to **79** characters. Since the ***DXF** format* supports only **31** characters you should limit layer names to **31** characters to guarantee a smooth ***DXF** Export*.

1. Select the name of the layer from the list box.
2. Type the new name.
3. Click the **Rename** button.

Deleting a layer

1. Select the name of the layer from the list box.
2. Click the **Delete** button.

   ***Caution**: The layer and everything on that layer is deleted; however, a warning appears before this happens.*

Making a layer the work layer

1. Select the name from the list.
2. Click the **Set Work** button.

   The layer must be visible before you can make it the work layer.

Using the layer box to specify a different work layer

You can use the **pop-up** layer box at the lower left of the drawing area to specify a different layer as the work layer.

1. Press the mouse button on the box and the menu displays.
2. ***Drag*** to the layer you want to be the work layer and release the mouse button.

Hiding a layer

You can't hide the work layer.

1. Click the layer(s) you want to hide in the list.
2. Click the **Hide** button.

   The visible mark beside the layer name disappears.

Displaying a hidden layer

1. Click the layer(s) you want to make visible in the list.
2. Click the **Show** button.

   A visible mark appears next to the layer's name in the list.

Determining how many objects are on a layer

1. Choose **Selection Mask** from the **Edit** menu.
2. Select the desired layer from those listed in the center box.
3. Choose **Select All** from the **Edit** menu.
4. Choose **Edit Objects** from the **Edit** Menu.

   The number of objects is listed at the top of the dialog box.
5. Close the **Selection Mask** and **Edit Objects** dialog boxes.

Using colors with layers

Layers facilitate color specific wiring diagrams or electrical and plumbing blueprints.

Many CAD drawings show different components in different colors. When you specify a different color for a layer, the layer ***remembers*** the color so that if you add geometry to a layer later, the new geometry appears in the same color as the last geometry constructed on that layer. In this way, you can use the **Visible** pen in **red** on one layer, **green** on another layer, and so on.

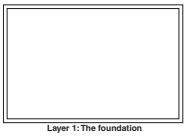
1. Select **Layers** from the **Layout** menu.
2. Create a new layer and rename it **Outline green**, indicating you always want to use the pen style **Outline** with a green color on this layer.

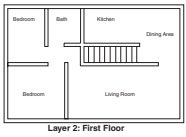
***Note:** Layers can be saved as Preferences along with specified colors.*

3. Make the layer **Outline green** current.
4. Select **Outline** in the **Style** submenu from the **Pen** menu.
5. Change the pen color from **Black** to **Green** by selecting **Green** in the **Color** submenu from the **Pen** menu.
6. Draw a circle.
   The circle is created with the green color.
7. Select the **Single line** tool.
8. Make **Layer 1** current.
9. Draw a line
   The line is created in black (the default color for the **Outline** pen style).
10. Select the **Rectangle** tool.
11. Make Layer **Outline green** current and draw a rectangle.
    The rectangle is created in **green**.

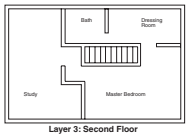
A Layer Example

Layers are like transparent sheets that you can turn on and off. This feature is particularly useful for designs with several variations of a component. A common use of layers is to draw the different floor plans and plumbing and wiring plans for a building on different layers. The following examples show simplified drawings using layers.


Layer 1: The foundation



Layer 2: First Floor



Layer 3: Second Floor

In order to print or plot the layers, simply use the **Layers** dialog box to turn on the layers you want to print and turn off all other layers. When you can see what you want to print on the screen, choose **Print** from the **File** menu.

Using Smart Walls with layers

You can also observe how smart walls function in these examples. The interior walls on **Layers 2** and **3** overlay the exterior walls of the foundation.

***Note**: Smart walls merge only on the same layer; therefore, the interior walls are not merged with the exterior walls and walls on **Layer 2** are not merged with walls on **Layer 3**.*

Layer Groups

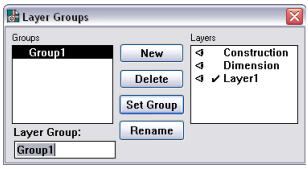
Layer Groups are particularly useful for viewing and plotting different layers of a complex drawings. For example, the document described in the **Layers** section that shows the floor plan of a house might have several layers illustrating the plumbing layout and several layers illustrating the electrical plan. The plumbing layers can be shown in the ***Plumbing*** *Layer Group* but hidden in the ***Electrical*** *Layer Group*. Similarly, the electrical layers are shown in the ***Electrical*** *Layer Group* but hidden in the ***Plumbing*** *Layer group*. To create a print to go to the electrical contractor, you can simply make the ***Electrical*** *Layer Group* be the work group. This will show the layout of the house with the electrical plan contained on various electrical layers but not the plumbing information which is on the hidden plumbing layers. Rather than having to **Show** or **Hide** numerous *individual* layers, you only have to select a single Layer Group to get the format that is desired.

Layer Groups

The **Layer Groups** command in the **Layout** menu extends the functionality of layers by creating *Layer Groups*.

A *Layer Group* records, stores, and displays layer settings—which layers are shown and which layers are hidden, as well as which layer is the work layer. This lets you show and hide different layers quickly by selecting a single *Layer Group* rather than setting the attributes of numerous individual layers.

When you select the **Layer Group** command from the **Layout** menu the following dialog box is displayed:



The **Layer Groups** dialog box lists the layer groups on the left and the individual layers on the right. Visible layers for the selected (highlighted) *Layer Group* are indicated in the **Layer** list by the **Eye Symbol**, just as they are in the **Layers** dialog box.

Only *visible* layers can be made the work layer within a *Layer Group*.

Visibility can be toggled by clicking the **Eye Symbol**, or at the location where the **Eye Symbol** should be for hidden layers.

The *work layer* of the selected group is set by double-clicking on the desired layer from the **Layer** list and is indicated by a checkmark.

Layers that are created after a *Layer Group* has been defined are included in the group but are not visible.

The current set of visible layers and the work layer need not correspond to any layer group; thus, it is not necessary that a *Layer Group* be designated as the "work" group.

Setting a work group simply changes the model's current layer settings to those of the specified *Layer Group*. Also, if a *Layer Group* is designated the *work group* and layer attributes are subsequently modified—either by the **Layer** dialog box or by changing the work layer in the **Layer Indicator** pull-down list—the *Layer Group* loses its "work" status.

A check mark by a *Layer Group* indicates that the current settings in the **Layer** dialog box match those of the indicated *Layer Group*. This is only valid just after displaying the **Layer Group** dialog box.

The **Layer Group** dialog box allows the following settings:

There must be at least one *Layer Group* created before you can use any of the buttons other than **New** or change any settings for any layers.

New This button creates a new *Layer Group*, called **Group1**, **Group2**, **Group3**, etc., by default. Use the **Rename** button to rename the Layer Group with an appropriate name. You can have up to 256 Layer Groups in a single file.

Delete This button deletes the selected (highlighted) *Layer Group*.

You can also activate a *Layer Group* by double-clicking the name of the *Layer Group*. The active *Layer Group* is indicated by a checkmark. When you open the **Layer Group** dialog and no *Layer Group* is indicated by a checkmark, the current work layer set in the **Layer** dialog box doesn't match the setting of the last activated *Layer Group*.

Set Work This button sets the currently selected (highlighted) group as the work group. This changes the layer settings for the current model to those specified by the layer group.

Rename This button renames the currently selected (highlighted) group with the name in the **Layer Group** entry box.

Creating a Layer Group

1. Click **New**. A new Layer Group appears in the group list. It records the current layer settings and displays these settings in the Layers list.
2. Change the layer settings as desired. *Visible* layers are indicated by the **Eye Symbol** and can be hidden by clicking on the symbol. *Hidden* layers can be made visible by clicking in the location where the **Eye Symbol** would be. The *work* layer is indicated by the **check mark** and can be set by double-clicking on the desired layer.

Renaming a Layer Group

1. Select the desired *Layer Group* from the **Group** list.
2. Enter a new name in the **Layer Group** entry box.
3. Click **Rename**.

Deleting a Layer Group

Deleting the work group, which is indicated by a **check mark**, deletes the group from the list, but the layer settings remain in effect until they are changed.

1. Select the desired *Layer Group* from the **Group** list.
2. Click **Delete**.

Making a Layer Group the work group

1. Select the desired *Layer Group* from the **Group** list.
2. Click **Set Group**.

   This changes the layer settings for the current model to those specified by the layer group. These settings are shown in the **Layers** list.

A check mark by a *Layer Group* shows the active *Layer Group* and indicates that the current settings in the **Layer** dialog box match those of the indicated *Layer Group*.

Changing the layer settings of a Layer Group

1. Select the desired *Layer Group* from the **Group** list.
2. Change the layer settings as desired.

   Visible layers are indicated by the **Eye Symbol** and can be hidden by clicking on the symbol. Hidden layers can be displayed in the layer group by clicking in the location where the **Eye Symbol** should be. The work layer is indicated by the **check mark** and can be set by double-clicking on the desired layer. The new settings automatically replace the old settings.

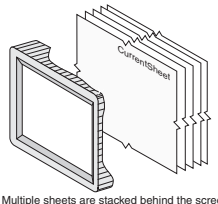
Sheets

A Sheet is an infinite planar area. ***DraftBoard*** allows you to have multiple opaque sheets within a drawing. A drawing can be a simple part or a complex assembly, and you can organize related geometry on as many sheets as the scope of the project calls for.

You can have as many sheets as you want but only one sheet can be visible at a time—the *active* sheet. You activate a sheet with the **Current** command in the **Sheets** dialog box in the **Views** menu.

In ***DraftBoard*** one sheet is always **Current**—the active sheet which cannot be deleted.

If you have multiple sheets, imagine them arranged as a stack of sheets behind your computer screen. If you make a sheet current you bring it to the front. That is why you can see only one sheet at a time—the **current** sheet.



Multiple sheets are stacked behind the screen. The current sheet is always in front and the only visible one.

Using Sheets

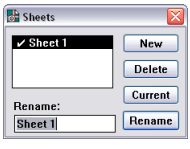
Sheets are ***DraftBoard's*** equivalent to pieces of paper you use with a drawing board.

Every document contains automatically **Sheet 1**.

Sheets

This command in the **Views** menu allows you to *create*, *delete*, *rename* sheets and set the *work sheet*. The checkmark (✓) in the list of sheets indicates the work (or current) sheet.

The name of the active sheet (*Document Name*: *Sheet Number*) is only displayed in the **Title Bar** if more than one sheet exists.



You may want to use several sheets to create related components and associated detail views. With the drafting board you use a new sheet of paper for a project; with ***DraftBoard*** you create a new sheet.

Whenever you want to print a part related to one project separately, create a new sheet.

Creating a new sheet

Click **New** to create another sheet. You can name the sheet by typing the name in the **Rename** entry box and clicking the **Rename** button.

You can also type a name and then click the **New** button.

Renaming a sheet

1. Select the name of the sheet from the list box.
2. Type the new name.
3. Click the **Rename** button.

Deleting a sheet

*Note: The current sheet **cannot** be deleted.*

1. Select the name of the sheet from the list box.
2. Click the **Delete** button.

When you delete a sheet you delete only the sheet including all *Detail Views*, but not the geometry displayed on the sheet.

Detail Views are described later in this chapter.

DraftBoard handles *Models* and *Sheets* separately. The geometry you create is the model—what you see on the sheet is only an image (view) of that model. So when you delete the sheet you delete only the **Sheet View** or **Detail View** of the model, but not the geometry (model) itself. You can display the deleted geometry any time on another sheet using the **Models** command in the **Views** menu.

See *Models* in the next section of this chapter for more information on how models work.

Making a sheet the current sheet

1. Select the name from the list.
2. Click the **Current** button.

Models

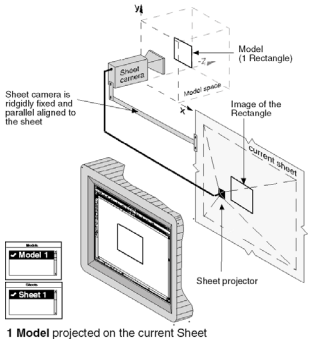
Models are composed of the geometry you are creating. Even if it looks like the geometry is created directly on the sheet, it is not. What you see on the sheet is only an image of that model. The model itself is created in an infinite three-dimensional area we call **Model space**.

Any combination of geometry such as lines, arcs, circles, dimensions, text, etc. you create with any of the creation tools in the *Tool palette* or any tools in the *Dimension palette* is a *Model*.

Projected Model Image

The image you see on the sheet is the projected **Sheet View** of the model picked up by the **Sheet Camera** which is aligned parallel to the sheet and looking from a top view, at the model.

It may be difficult to imagine that when you draw in ***DraftBoard*** two-dimensional geometry, that this geometry is created in a three dimensional **Model Space**. It is easier to think of that model space as a thin pane of glass where all geometry is created. An image of everything that is placed on that pane of glass is continuously projected on the current sheet.



1 Model projected on the current Sheet

When you use the **Zoom** commands in the **Arrange** menu or the **Zoom** tools from the *Tool palette* you change the view scale of the model by zooming the *Sheet Camera* like you would with a video camera.

Multiple Models

There are two different ways to create models:

- with the **Sheets** command in the **Views** menu

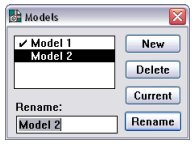
  or

- with the **Model** command in the **Views** menu.

When you create a new sheet, ***DraftBoard*** automatically creates a new blank model where you can create geometry.

Models

This command in the **Views** menu allows you to specify models and switch between models to project them on the current sheet. You can display a single model on more than one sheet, but you can only see one model at a time on each sheet without using detail views.



Creating a new model

You can also type a name and then click the **New** button.

Click **New** in the **Models** dialog box to create another model. You can name the model by typing the name in the **Rename** entry box and clicking the **Rename** button.

Renaming a model

1. Select the name of the model from the list box.
2. Type the new name.
3. Click the **Rename** button.

Making a model the current model

1. Select the name from the list in the **Models** dialog box
2. Click the **Set Work** button.

Deleting a model

1. Select the name of the model from the list box.
2. Click the **Delete** button.

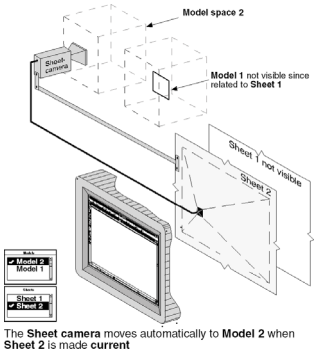
When you delete a model, you delete all of the geometry making up that model.

*Note: You **cannot** delete the current model. A model which is still associated to a sheet cannot be deleted. As long it is related to a sheet the **Delete** button in the Models Dialog box is greyed out.*

Relationship between Sheets and Models

When you create a new model with the **Model** command, a new model is created without creating a new sheet. So it is possible to have more models than there are sheets. You can only look at one model at a time in any view *(sheet view* or *detail view*).
So it is also possible to have a model that is not displayed in any view. And you can have a single model displayed in many views at the same time.

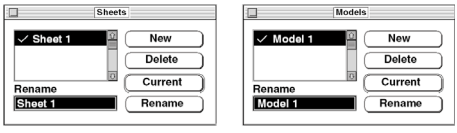
When you make a Model current in a sheet view, it is like the **Sheet camera** is moved to the new model. When you make a **Sheet** current, the **Sheet camera** is moved to the related model whose image is projected on that sheet.



The **Sheet camera** moves automatically to **Model 2** when **Sheet 2** is made **current**

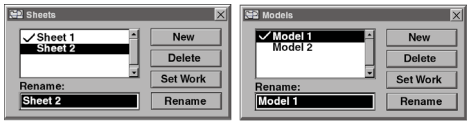
Creating a new Model with the Sheets command

1. Select **Sheets** in the **Views** menu.
   The **Sheet** Dialog box displays.
2. Select **Models** in the **Views** menu.
   The **Model** Dialog box displays.
3. Since the **Model** dialog box is placed at the same location on the screen, move it to another location so you can see both dialog boxes.



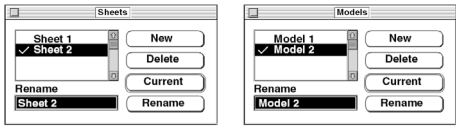
Default Setup after launching **DraftBoard**

   Now you can see that a new drawing in ***DraftBoard*** has one **Sheet** and one **Model** by default. The checkmark in each dialog box shows that **Sheet 1** and **Model 1** are current.
4. Display the **Sheets** Dialog box and click **New** to create a new sheet.
   You see also that a new model automatically appears in the **Models** Dialog box (**Model 2**).



Adding a new Sheet automatically creates a new Model

5. Make **Sheet 2** current by highlighting it and clicking **Current**.
   If you have any geometry on **Sheet 1**, it disappears and a blank screen comes up. Also, the check mark moves from **Sheet 1** to **Sheet 2** and the check mark also moves automatically from **Model 1** to **Model 2**.



Making **Sheet 2** current automatically makes **Model 2** current

Model 2 automatically becomes current because **Sheet 2** is related to **Model 2**. So if you make **Sheet 2** current on the screen, then **Model 2** has to be current as well.

Sheet 2, now displayed on the monitor, is blank because nothing has yet been added to **Model 2**. As soon as you begin to draw, all geometry is automatically added to **Model 2** since **Model 2** is the current model space.

Associative Views

Views allow you to enlarge or reduce all or part of the geometry for both viewing and printing. This visual change of the view does not affect the actual measurement of the geometry (the model). There exist two types of views in ***DraftBoard***: the **Sheet View** and **Detail Views**.

Sheet View — The *Sheet View* shows the object geometry on the current sheet at the scale you set with the **Drawing Size** command in the **Layout** menu. The *Sheet View* is the default view when you launch ***DraftBoard***. The *Sheet View* is picked up by the *Sheet camera* and projected on the current sheet. The *Sheet View* is infinite. It has no boundaries.

Detail Views — A detail view is created with either the **Detail View** tool on the tool palette or by the **Sheet Into View** command in the **Views** menu. **Detail Views** are always displayed in a view window.

These two different views are easily to identify on the screen:

- Each item displayed **inside** a view window is a **Detail View**.
- The area **outside** (and underneath) all detail view windows is the **Sheet View**.

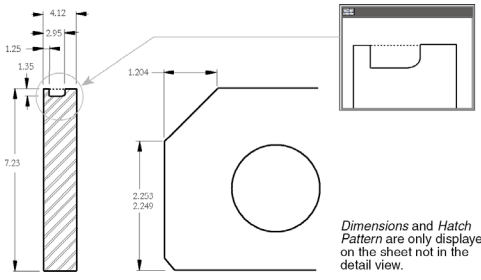
Creating Detail views using the Detail View tool

DraftBoard allows you to create *associative Detail Views*. These *Detail Views* are always placed in a view window and are associative so that when you modify geometry in any view, all views of that geometry reflect the change. There are three deliberate exceptions to this associativity, in line with standard drafting practice:

- Dimensions
- Crosshatching
- Text

They are *view dependant* and appear only in the view in which they were created.

In the illustration below, the detail view shows a **2 : 1** enlargement of the side notch. *Dimensions*, *text* and *hatches* are displayed only on the sheet where they were created.



View Windows can be created in ***DraftBoard*** in two different ways:

- manually with the **Detail View** tool

  or

- automatically with the **Sheet Into View** command in the **Views** menu.

Regardless of the method you use to create a **View Window**, they all behave the same way.

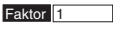
Detail View Tool

The **Detail View** tool on the **View Control** subpalette of the *Tool Palette* creates a detail view of the designated area of your drawing. The detail view is *associative* so that when you modify the geometry, the detail view reflects the change

How to create a *non-associative* detail view is described at the end of this chapter.

Creating an associative detail view

1. Select the **Detail View** icon from the **View Control** subpalette.
2. Enter a **Scale** for your detail view in the status line.

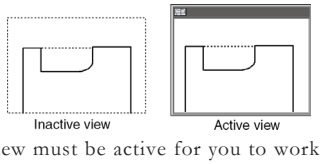
   

3. Use the pointer to ***drag*** a rectangle around the area of your drawing that you want placed in the detail view. This rectangle becomes the window frame.
4. Position the pointer in the center of the detail view window frame and ***drag*** the window to a clear area on your drawing.

You may also create the rectangle by clicking to place the diagonally-opposite corners.

Activating/deactivating detail views

A *Detail View* is made *active* by clicking inside the view window. An active detail view displays its title bar and **Detail View** pull-down menu.



When a view window displays a title bar the view window is active. When the view window has no title bar, the underlying sheet is active.

A view must be active for you to work in it. Either a detail view is active or the sheet view is active, but only one can be active at a time.

You deactivate a *Detail View* by clicking in the drawing area away from all detail views. In order to draw on the sheet it must be active.

Placing a Detail view on a separate Sheet

1. Create a detail view on the current sheet.
2. Select the **Cut** command in the ***pull-down*** *menu* of the detail view.
3. Choose the **Sheet** command in the **Views** menu and create a new sheet (**Sheet 2**) using the **New** command in the **Sheet** dialog box.
4. Active **Sheet 2** by clicking the **Current** button.
5. Choose the **Insert** command in the **Edit** menu to place the detail view onto **Sheet 2**.

   The detail view is placed on **Sheet 2** and is associatively connected to the object geometry on **Sheet 1**.

Don't use the **Cut** command in the **Edit** menu, since it cuts the geometry and not the detail view.

Flattening Views

The two commands **Flatten View** and **Flatten all Views** project all geometry visible in a detail view window onto the current sheet. The command **Flatten View** projects only one view window (the active view), while the command **Flatten all Views** all views projects.

Flatten View

*Note: All geometry invisible will not be projected but cut and deleted. Therefore you must use the **Zoom all** command if you want to project all geometry in a view window.*

This command in the **Views** menu allows you to remove hidden three-dimensional geometry in direction of the ***Z**-Axis* in a view window .

The command **Flatten View** in the **View** menu places all geometry visible in a view window with the current scale factor onto the current sheet and removes overlapping lines of equal length and all lines parallel to the view direction. The object geometry is no longer associative to the related model. When you edit a flattened view the related model remains unchanged and vice versa.

Flatten a View

1. Select a view window.
2. Choose the **Flatten View** command in the **Views** menu.

   A dialog window is displayed warning you that the selected view will be disassociated from the related model.

   All dimensions in the view window will be deleted, since three-dimensional dimensions would not be correct in a two-dimensional drawing.
3. Click **OK**.

   The object geometry will be placed with the current scale factor onto the current sheet.

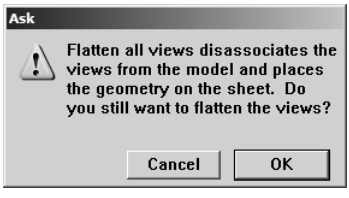
Using the **Flatten View** command *Text* and *Hatches* will be handled such as object geometry. Be aware, that dimensions are not any more automatically adjusted in the view window when you edit the model, since the model is disassociated from the detail view. Dimensions in the flattened view are only automatically adjusted when you edit the flattened object geometry.

Flatten all Views

Often **Detail views** and the **Sheet view** show different models. When exporting drawings only one model can be exported, therefore detail views can get lost. This can be avoided with the **Flatten all Views** command.

When exporting ***DXF/DWG** Files* not all views have to be flattened, since several views are automatically detected during export and a dialog box with corresponding options will be displayed. See as well *Exporting **DXF/DWG**-Files* in this manual.

When you select this command the following dialog window is displayed:

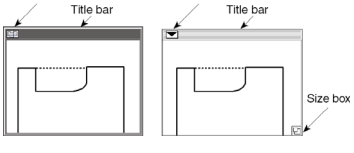

Ask
Flatten all views disassociates the views from the model and places the geometry on the sheet. Do you still want to flatten the views?
Cancel
OK

This note warns you that all detail views will be disassociated from the model and therefore will loose their associativity.

When exporting a drawings with more than one model, using the export formats **EMF**, **WMF**, **PNG**, **DXF/DWG** you will automatically be offered to flatten all views if you want to export all detail views.

View Window

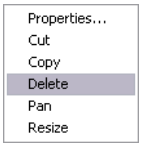
The view window includes a *Title bar*, *Size box* and **pull-down** menu only when the detail view is active.



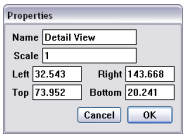
The left graphic shows a *Detail View window* for *Windows*, the right graphic for the *Apple Macintosh*.

You can ***drag*** the view around by the title bar and resize the window by ***dragging*** the size box.

The **Detail View** menu provides options for manipulating the view.



Properties The **Properties** option allows you to change the view **scale** and define the locations of its corners.



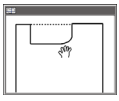
Cut The **Cut** option allows you to remove the view from the sheet and place it on the *Clipboard*. This is useful for placing views on a different sheet. Use the **Paste** command from the **Edit** menu to paste the cut view window.

Copy The **Copy** option allows you to place a copy of the view window on the *Clipboard*. This is useful for pasting multiple identical views onto one or more sheets. Use the **Paste** command from the **Edit** pull-down menu to paste a copy of the view window.

Delete This option deletes the active window and makes the sheet active. It does not delete geometry.

Pan This command allows you to move the geometry inside the view window. A hand icon appears which you can use to ***drag*** the contents of the window.

When you release the mouse button, the **Pan** function ends.



Resize The **Resize** option allows you to ***drag*** a new view window. This function differs from dragging the window borders. With the **Resize** command, you do not have to change each border individually since you define a new window that is substituted for the original one.

Changing the view scale

When you are trying to fit a view or group of views in a drawing format, you may have to scale a view.

1. Choose **Properties** from the **Detail Views** menu.
2. Specify the scale you want and click **OK**.

Zooming within a view

1. Click inside the view to make it active.
2. Hold down the **Shift** key (*Macintosh:* ***Ctrl*** *key*).
3. Choose the **Zoom** command you want from the **Arrange** menu.

   The geometry inside the view is zoomed.

Scrolling within a view

1. Click inside the view to make it active.
2. Hold down the **Shift** key (*Macintosh:* ***Ctrl*** *key*) and move the scrollbars.

   The geometry inside the view is moved.

10

All commands from the **Arrange** pull-down menu work inside the view when you hold down the **Shift** key (*Macintosh:* ***Ctrl*** *key*) while choosing from the menu.

All scrollbar functionality can be used inside the view when holding down the **Shift** key (*Macintosh:* ***Ctrl*** *key*) while moving the scrollbars.

Bring To Front

This command in the **Views** pull-down menu brings the specified view to the front of the **stack** of views.

Bringing a specified view to the front

1. Select the view.
2. Choose **Bring To Front** from the **Views** menu.

This command counteracts the **Auto Front** command, so when you click, you select objects in the view rather than activating another view.

Send To Back

This command in the **Views** pull-down menu sends the specified view to the back of the **stack** of views.

Bringing a specified view to the back

1. Select the view.
2. Choose **Send To Back** from the **Views** pull-down menu.

This counteracts the **Auto Front** command, so when you click, you activate another view rather than selecting objects.

Auto Front

This command only works for views.

The **Auto Front** command in the **Views** menu overrides the **Bring To Front** or **Send To Back** settings. When this setting is in effect, you can simply click on a view to bring it to the front and make it active.

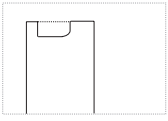
Handling overlapping views

The view control commands in the **Views** menu allow you to control the active view. When you have overlapping views, you can think of them as a stack where only the top view can be active. If the **Auto Front** command is turned on and you click a view it comes to the top of the stack and you can work in it. This may cause some problems when you click an object in the top view to select it, but an underlying view is activated instead.

To remedy this situation, turn off the **Auto Front** command. Select the view you want to work in and choose **Bring To Front** from the **Views** menu. The objects in the view you brought to the front remain on top, even when you click a different view visible in the top view's space.

Draw View Boundaries

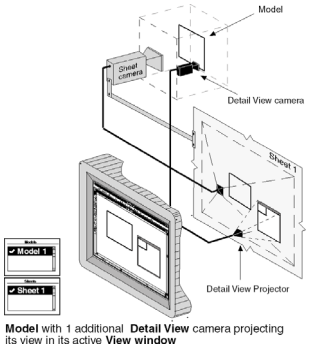
This command in the **Views** menu shows the boundaries of all inactive views. This boundary will be printed if you leave it displayed when you choose the **Print** command. If you want to print the view without boundaries deactivate **Draw View Boundaries** in the **Views** menu.



How Detail Views work

When you create a detail view with the **Detail View** tool on the Tool palette or the **Sheet Into View** command in the **Views** menu, you see the same model through a second camera—the **Detail View Camera**.

The **Detail View** camera behaves like the **Sheet Camera** except it displays its view of the model in a **View window** on the sheet.



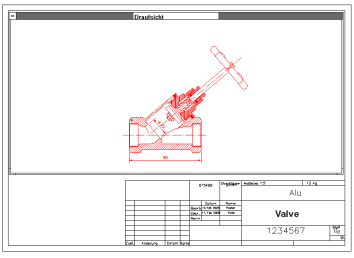
Model with 1 additional **Detail View** camera projecting its view in its active **View window**

All views displaying the same model are associative. For example, when you change the geometry on the sheet, the geometry in the view window changes, and vice versa.

When you have several views displaying the same model they have to be associative, because no matter which view is active—the **Sheet View** or any **Detail View**—you are always editing the same model.

Creating Views using the Sheet Into View command

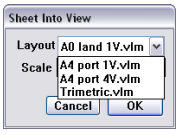
The **Sheet Into View** command places all the geometry on a sheet **including** *dimensions*, *text*, *hatching*, and *fills* into one view window (the front) at the scale you specify.



Sheet Into View

When you choose the **Sheet Into View** command the following dialog box is displayed:

In the **Layout** list you find as well *Design Layouts* called as **Design 3** or **Draft 4**. *These Design Layouts* are only used for designing in 3D. More about *Design Layouts* you find in the *3D Section* of this manual.



The **Sheet Into View** dialog box allows the following settings:

Layout — In this ***pull-down*** *list* you can select a paper size with view(s) in a standard drawing format.

Scale — In this entry field you can enter a scaling ratio, such as **1: 8**, a size ratio such as **1cm:1m** or a factor. Factors greater than **1** enlarge all objects in the views and factors less than one reduce all objects.

Cancel — This button closes the **Sheet Into View** dialog box without executing the **Sheet Into View** command.

OK — When you click the **OK** button, all geometry on the sheet **including** *dimensions*, *text*, *hatching*, and *fills* is placed into one or several view windows at the scale you specify.

You can add your own formats to this list or modify the layouts provided to meet your needs is described in the next section.

Using Sheet Into View

1. Draw the geometry
2. Delete all existing *Detail View windows*.
3. Choose **Sheet Into View** from the **Views** menu.
   A dialog box is displayed.
4. Choose the drawing format and the number of views you want from the **Layout** list.
5. Enter the scaling value in the **Scale** box.
6. Click **OK**.
7. Select **Zoom All** in the **Arrange** menu to see the results.

Important: If you want to keep the existing detail views you must import the drawing format.

The geometry is scaled and the views are layed out as specified. If you specify a drawing format, it is imported into the view and scaled appropriately for the drawing size you have specified with the **Drawing Size** command in the **Layout** menu.

Scaling Views

The **Scale** you specify in the **Sheet Into View** dialog box changes the view scale property of that view only in relation to the object geometry on the sheet. It can be modified using the **Properties** command in the **Detail View** menu or using the **Zoom** tools.

When you use the **Drawing Size** command in the **Layout** menu to scale the entire drawing for printing the specified relation ship between the *Detail View* and the *Sheet* remains unchanged.

Recovering from an erroneous Sheet Into View

If you mistakenly invoked the **Sheet Into View** command and do not want your geometry in a view, use **Undo** to recover. If this is not done immediately, you must use the following method to manually back out.

1. Select all object geometry including *text*, *dimensions* and *hatch patterns*.
2. Choose the **Group** command in the **Arrange** menu.
3. Delete all view windows on the current sheet by choosing **Delete** from the *View window* menu of each view.

You can undo a **Sheet Into View** command using the **Undo** command in the **Edit** menu. But the **Undo** command works only if it is invoked immediately after the **Sheet Into View** command.

4. Make **Model 1** in the **Model** dialog box the current model for the sheet.
5. Delete **Model 2**.

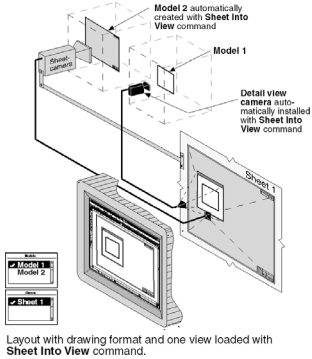
How the Sheet Into View command works

The view window created with the **Sheet Into View** behaves like any view window created with the **Detail View** tool.

Although both view windows behave the same, the **Sheet Into View** includes three automatic steps which are not performed when you create a view window with the **Detail View** tool.

The **Sheet Into View** command

- creates an additional **Model** (blank or containing a standard drawing format if selected).
- moves the **Sheet camera** to that model (blank or with the drawing format) to display it on the current sheet
- puts all geometry ***including*** *text*, *dimensions*, *hatching*, and *fills* on the current sheet into one or several view windows and displays it at a specific scale within the drawing format you have selected.



Layout with drawing format and one view loaded with **Sheet Into View** command.

You can add your own formats to this list or modify the layouts provided to meet your needs is described in the next section.

Drawing Formats

You cannot only modify all drawing formats in the **Layout** folder of the **DraftBoard** directory but create as well your own drawing formats and design layouts.

Drawing formats contain the *border line* of the drawing, the *cutting line* and a *Title block* for entries such as *name* or *scale* of the drawing.

Drawing formats have to be created for each paper format such as *A*, *B*, *C* or *D* and for both paper orientations *Portrait* and *Landscape*.

All drawing formats you find in the **Layout** folder of **DraftBoard** are designed for *Plotter devices*, where the *Cutting lines* correspond exactly with the dimensions of the selected paper format (for example *30 x 40 inch* for the *D* Format) and the *Border line* is offset inside by *1/2 inch* according to the ANSI standard.

How you create your own drawing formats for your laser printer is described later in this chapter.
In the **Layout** folder you find as well *Design Layouts* called **Design 3** or **Draft 4**. These *Design Layouts* are only used for designing in 3D. More about *Design Layouts* you find in the *3D Section* of this manual.

When you want to use these plotter formats for your laser printer you have to adjust them to the printing area of your printer, that could differ from printer to printer.

The *Standard Drawing Formats* shipping with ***DraftBoard*** contain in addition to the drawing format *detail views* that may display with the **Sheet Into View** command the content of a drawing at different view angles and scale factor.

Placing Drawing formats

There are two different ways to place *Standard Drawing Formats*:

- with the **Import** command in the **File** menu

  or

- with the **Sheet Into View** command in the **Views** menu.

Since placing a drawing format using the **Sheet Into View** command is described in the previous section the following task describes how that is done using the Import command.

Importing Drawing Formats

1. Scale the drawing using the **Drawing Size** command in the **Layout** menu.
2. Mark the option **Always display Page bounds** in the displayed **Drawing Size** dialog that allows you to compare the true size of the drawing with the displayed printing area of the drawing format.
3. Activate the current sheet by clicking on it with the **Selection tool**, in case your drawing contains any detail views.

   The window boundaries are displayed by *dotted lines* to indicate that the next step is executed on the current sheet and not within a detail view.
4. Choose the **Import** command of the **File** menu.

   The **Import** dialog box is displayed.
5. Open the folder **Layout** and select the drawing format you want.
6. Click the **Open** button.
7. Mark the option **unscaled** in the displayed dialog box.
8. Click the **OK** button.

   The *Drawing Format* is imported onto the current sheet. The imported drawing format is *grouped*.
9. Now you can ***drag*** the drawing format on the sheet until all objects lie within the drawing format.

In case the size of the imported drawing format is to small or to large to frame the entire drawing you can scale the content of the drawing.

The **Drawing size** command displays the printing area of the selected paper size at its true size on the drawing area. You can change the paper format using the **Print Setup** command (*Macintosh:* ***Page format***) in the **File** menu.

You can ignore the number of detail views since they are only important when using the **Sheet Into View** command.

Modifying Drawing Formats

All drawing formats provided in the **Layout** folder can be modified to meet your needs. When you want to add new formats is recommended to modify existing ones and save them under a different name.

Creating Drawing formats for Laser printers

1. Open a drawing format in the **Layout** folder of the **DraftBoard** folder.
2. Save the drawing under a new name.
3. Select the **Drawing Size** command in the **Layout** menu.

   The **Drawing Size** dialog box is displayed.
4. Mark the option **Display always Page Bounds**.

   The *Printing area* of your active Laser Printer is displayed as a grey rectangle.
5. Click **OK**.

   The **Drawing Size** command is closed.
6. Mark the grouped drawing format on the drawing area and ungroup with the **Ungroup** command in the **Arrange** menu.
7. Adjust the *border lines* of the drawing format until they are exactly aligned with the displayed grey rectangle for the printing area of your laser printer.
8. Delete all *cutting lines* since they are not required for print outs on a laser printer.
9. In case you want only to import the modified drawing format and not to use it with the **Sheet Into View** command you may delete the existing detail views.

   To delete a detail view by selecting the **Delete** command in the menu of the detail view.
10. Hide the **TitleBlocks** Layer.

    All *Text Entries* are hidden.
11. Select the complete drawing format using the **Select all** command in the **Edit** menu and group the format using the **Group** command in the **Arrange** menu.
12. Activate the layer **TitleBlocks**.
13. Save your work.

Important*: If the rectangle is displayed vertically you have to change the paper orientation with the* ***Print Setup*** *command (Macintosh:* ***Page format****) in the* ***File*** *menu and call the* ***Drawing Size*** *command again.*

Tip*: It is recommended to keep at least one detail view in a drawing format, since that allows you to use this format with the* ***Sheet Into View*** *command. The* ***Import*** *command ignores all detail views.*

Important*: Text entries with an @ character in front must not be grouped since then the* ***TitleBlocks*** *command cannot identify these entries as title entries (See as well under* ***AutoText*** *in this Manual).*

Adding formats to the Sheet Into View dialog box list

You can create your own drawing formats and add them to the list in the **Sheet Into View** dialog box.

Drawing formats that should be used with the **Sheet Into View** command must have at least *1 Detail View* into which all object geometry can be placed. Without any detail view the **Sheet Into View** command doesn't work correctly.

You can as well import a drawing formats containing one or several detail views, since the **Import** command ignores all detail views.

1. Start a new document.
2. Select the **Detail View** tool and create a view window.
3. Display the **Model** Dialog box and create a new model.
4. Click on the sheet to activate it.
5. Select **Model 2** in the **Model** dialog box and make it the current model by clicking the **Current** button.
6. Click in the view window to activate it and make **Model 1** the current model in the **Model** dialog box.
7. Click on the sheet to activate **Model 2**.
8. Create the drawing format at a scale **1:1** or import one of the standing formats and modify it.
9. Save the drawing with a proper name in the **Layouts** folder of ***DraftBoard***.

   The file name is listed in the **Layout** list field when you next time select the **Sheet Into View** command.

Combining Sheets, Views and Models

Sheets, *Views* and *Models* are directly related to each other and are interacting permanently unnoticed from the user.

For most of your daily work, it is not necessary to know anything about the relationship between *Sheets*, *Views* and *Models*. But for some tasks it is helpful to understand this relationship since it offers elegant solutions that were not possible without the combined power of this features.

Using the relationship between *Sheets*, *Views* and *Models* makes it easy to:

- create detail views of an object that is not associative to the original model.
- create customized drafting layouts or
- recover from an erroneous **Sheet Into View** command.

The first section of this chapter explained the use of *Sheets*, *Views* and *Models*. Each is a simple and straight forward operation. In the graphics you could see the environment, how *sheets*, *views*, and *models* are set up in ***DraftBoard***. What you still need to know is the exact definition of all engaged components and the rules that describe how they are interacting.

Virtual Design Environment

The environment in ***DraftBoard*** for handling all geometry you create can be described by five components:

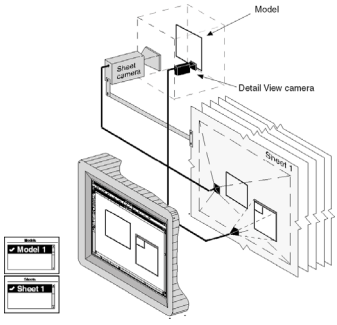
- Sheets
- Models
- Views with
  - Cameras and
  - Projectors.

First, you have to know that all geometry you create is not created directly on the sheet you see on your computer screen but somewhere outside of the sheet in an infinitely large three-dimensional work space (**Model space**). All geometry is *placed here as separated models. You only see on your sheet images, or views of these models.

Sheets

A sheet is an infinite 2D planar area that displays an image of one or more models. The image of a model is picked up either by the *Sheet camera* and projected onto the sheet (the *Sheet View*) or by a *Detail View camera* and projected into a view window which is resting on the sheet (the *Detail View*).

- Sheets are arranged behind each other so you can see only one at a time.



- You can create as many sheets as you want, but with each sheet you automatically create a **blank** model, similar to the drafting board when you start with a **clean** sheet.
- Independent of how many models exist, you can delete all sheets but one—the current sheet—since ***DraftBoard*** needs at least one sheet to display the models even if they are blank.
- When you delete a sheet, all *Detail Views* resting on that sheet are deleted.
- You can display as many *Detail Views* as you want on one sheet but only one sheet (*Sheet View*) at a time.
- When you change the current sheet, the related model is always activated. You can change this relation only by assigning another model to that sheet.

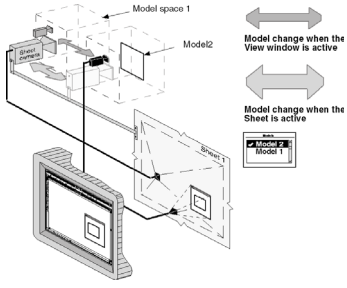
On the drawing board you start a new drawing with a blank sheet like in *DraftBoard*. Only *DraftBoard* needs a ***blank*** *Model* to display a ***blank*** *Sheet view*.

Models

A Model is a collection of geometry, dimensions, text, fills and hatching. Models are placed in an infinite three-dimensional area. A model can be blank (then we call it *Model space*) just as it is when you launch ***DraftBoard*** and look at a blank sheet. Images of models are picked up by either the *Sheet camera* or by *Detail View cameras* and projected on the sheet. The view of the *Sheet camera* is called a *Sheet View* and the views of the *Detail cameras* are called *Detail Views*.

- You can create and delete as many models as you want.
- Models exist independent from the sheets and views.
- When you change the current model, you move the related camera. The camera you move depends on what is active—the sheet or the detail view. If the sheet is active, the *Sheet camera* is moved to the current model, if the detail view is active then the related *Detail view camera* is moved.

You change the current model by selecting a model in the **Models** dialog box and click the **Current** button.



Views

Views are the images picked up by cameras and projected on sheets. ***DraftBoard*** has two types of views—the *Sheet View* and *Detail Views*.

Detail Views

- *Detail Views* are picked up by *Detail View cameras* and always displayed in a view window which rests on the sheet. They have boundaries and display only a limited view of the model.

  You can create as many views as you want and move or copy and paste them to different sheets.

  When you delete a *Detail View* the *Detail View camera* is deleted. When you delete a sheet which contains a *Detail View*, the *Detail View* is deleted also since the *Detail View* is resting on the sheet.

- You can place views from different models on one sheet, but each view can display only one model at a time.

Sheet View

The **Drawing Size** command changes only the plot scale but not the scale factor of the Sheet view. Even if the view is scaled to fit the paper format the scale factor remains **1:1** since the real size of the geometry is not affected.

- There is only one *Sheet View* for each sheet. The *Sheet View* is an infinite view picked up by the *Sheet camera* and displays everything on the sheet outside of all view windows.

  The *Sheet View* cannot be deleted and needs at least one sheet to display its view.

  Since the *Sheet View*, like the *Detail View*, can display only one model at a time, you have to use *Detail Views* to show more than one model on a sheet.
- To activate a *Detail View* you have to click in the view window, to activate the *Sheet View* you have to click on the sheet outside of all detail views. If the model dialog box is displayed, the related model will be highlighted.

Cameras and Projectors

There are two type of cameras—one *Sheet camera* and as many *Detail View cameras* as you create detail views.

- The Sheet camera is permanently installed as default and displays its view via the sheet projector on the sheet.

  To move the *Sheet camera* you have to perform a model change either by the **Models** command or by the **Sheets** command where the *Sheet camera* is moved automatically to the related model.

When you change the sheet the *Sheet camera* is moved to the selected sheet and automatically displays it at the view scale which was current when the sheet was last changed.

- *Detail View cameras* display their views via *Detail View projectors* in view windows which rest on the sheet.

  You can install as many *Detail view cameras* as you want even if each one is looking at one model. You install *Detail view cameras* by creating *Detail views* either by the **Detail View** tool from the Tool palette or with the **Sheet Into View** command.

  You can move *Detail View cameras* to another model by performing a model change when the view window is active.
- Both cameras, the *Sheet camera* and a *Detail View camera* can look only at one model at a time.

Using Sheets, Views and Models

We listed a lot of rules in the sections above and they all describe in common the same fact that ***DraftBoard*** administrates *geometry* (models), *views* and *sheets* separately. That has a lot of advantages. The most impressive example is if you delete a sheet displaying geometry the geometry is not lost since you have not deleted the related model.

The following examples show how to use the combination of *Models*, *Sheets* and *Views* for your daily work.

Recovering from deleting a sheet displaying geometry

To restore the geometry displayed on a sheet you deleted unintentionally, proceed as follows:

1. Select **Sheets** in the **Views** menu.
2. Click **New** to create a new sheet.
3. Click **Current** to activate the new sheet and draw some geometry.
4. Make another sheet active by highlighting a different sheet in the **Sheets** dialog box and click the **Current** button.
5. Close the **Sheets** dialog box and select **Models** in the **Views** menu.
6. Search for the before-displayed geometry by activating each *Model* followed by a **Zoom All** command until the previous geometry is displayed on the new sheet.

Identifying Models

This procedure is very useful if you have to identify models and sheets for renaming.

If you do not know which view is displaying which model, open the **Models** dialog box. When you click on the drawing area, the check mark moves to the model displayed on the sheet. If you click in a view window, the check mark moves to the model displayed in the detail view.

Creating a non-associative detail view

You may want a detail view that is not associative; that is, when you make changes to the original geometry, the geometry in the view does not change.

1. Create a detail view on an open space containing all geometry you want to be non-associative.
2. Use the **Selection** tool to click inside the view to activate it.
3. ***Drag*** a selection marquee around all of the geometry inside the original view.

   The geometry inside the view is selected.
4. Choose **Copy** from the **Edit** menu.
5. Choose **Models** from the **Views** menu.
6. Click the **New** button to create a new model (by default, model **2**).
7. Click the **Current** button to make the new model the current model in the new view.

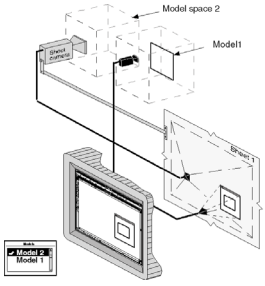
   The geometry in the detail view disappears.
8. Choose **Paste** from the **Edit** menu.

   The geometry is pasted into the new detail view, but it is no longer associative, since it is part of a different model.

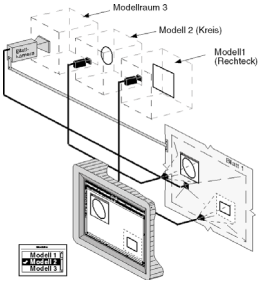
Displaying several components on one sheet.

Sheets, *Views* and *Models* are very helpful to display several components of a part created by different drafters on one sheet.
The following simple example shows how to perform this task:

1. **Open** a new document.
2. Draw an ellipse.
3. Save the drawing as **Ellipse.vlm**.
4. Open a new document and draw a rectangle.
5. Select **Sheet Into View** in the **Views** menu.
6. Choose the Layout **Top View**, set the Scale to **1** and click **OK**.



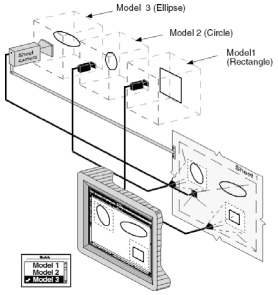
   You now have one **Sheet** with one view window showing the Rectangle (**Model 1**). The Sheet camera is aligned to **Model 2** (created with the **Sheet Into View** command) displaying a blank model space on the sheet.
7. Make the sheet active by clicking on it and draw a circle.
8. Select **Sheet Into View** (the sheet still active) and choose **Top View** again with the **Scale** factor **1**.



Now you have one Sheet with two view windows, one displaying the rectangle (**Model 1**) and one displaying the circle which became **Model 2**. The Sheet camera is moved to the new **Model 3** and an empty model space is displayed on the sheet.

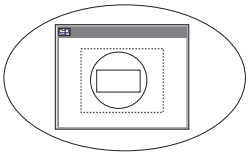
9. Activate the sheet by clicking on it and select **Import** in the **File** menu.

10. Select the file **Ellipse.vlm** and click **OK**.
    The ellipse is imported on the current sheet and added to **Model 3** which was active when you performed the **Import** command.

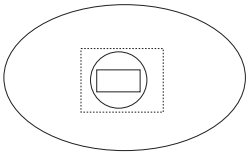


11. Arrange the two view windows around the ellipse and rearrange the rectangle and the circle with the **Zoom** commands in the **Arrange** menu like in the following graphic.

In order to select overlapping view windows, **Auto Front** must be deactivated, as it is described in the *Viewing Geometry* chapter.



12. Activate the sheet and deactivate the command **Show View Boundaries** in the **Views** pull-down menu.



CHAPTER

11

DraftBoard Documents

Handling Documents

Data Exchange

DraftBoard Documents

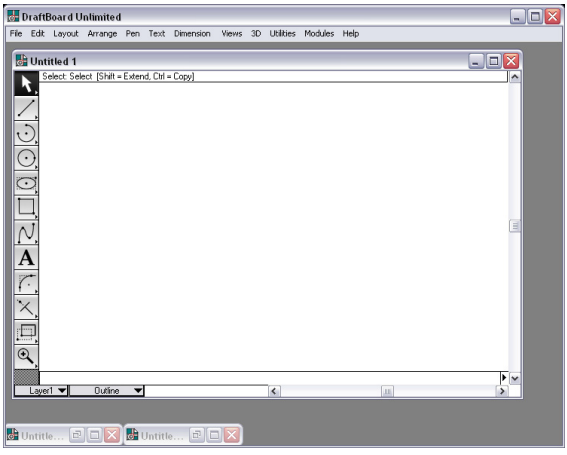
This chapter describes the handling and administration of *Files* (in ***DraftBoard*** called drawings) and the data exchange with other programs.

Using Documents

A document is a ***DraftBoard*** file. Whenever you open ***DraftBoard***, or choose **New** from the **File** menu, a new document appears in its window.

Working with multiple Documents

In ***DraftBoard*** you can open and edit several drawings simultaneously. Each *Drawing* is displayed in its own window that can be reduced to an icon near the lower-left corner of the ***DraftBoard*** work space. To close, move or resize a drawing you use the control buttons in the title bar of each window.

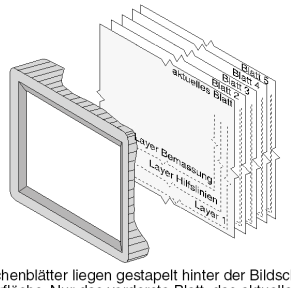

DraftBoard Unlimited
File Edit Layout Arrange Pen Text Dimension Views 3D Utilities Modules Help
Untitled 1
Select: Select (Shift = Extend, Ctrl = Copy)
Layer1
Outline

Windows

The **Windows** submenu in the **Views** menu shows the names of all the open ***DraftBoard*** *documents*. To bring a document to the front, that is covered by another document, select its name from the **Windows** submenu.

Components of a Document

A drawing can be a simple part or a complex assembly. Each *Drawing* has one or more *Sheets*, like the sheets of paper in a set of blueprints, and each *Sheet* is made up of *Layers* which can be hidden or displayed as needed.



Zeichenblätter liegen gestapelt hinter der Bildschirmoberfläche. Nur das vorderste Blatt, das aktuelle Zeichenblatt, ist sichtbar. Jedes Zeichenblatt kann wiederum bis zu 256 Layer besitzen.

The first part of this chapter discusses using the **File** commands for *opening*, *importing*, *exporting* and *saving* files. Printing and plotting documents is described in the *Basic Drafting* chapter.

Document Properties and Preview

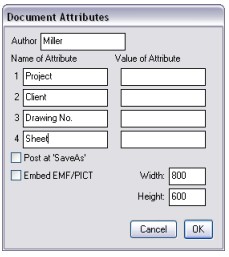
For each drawing you can save specific document properties and a preview of the drawing, that can be optionally displayed in the *Windows* ***Open*** dialog box and in the ***Windows*** *Explorer* (**Win98**, **WinSE**, **WinME**, **Win2000**, **WinXP**).

All these attributes are set with the **Documents** command in the **Preferences** submenu in the **Layout** menu.

Document Properties

This command allows to define document properties and specify if these values should be displayed when saving a drawing. In addition you can specify, if a preview of the drawing should be saved with the document.

When you choose this command the following dialog window is displayed:



The **Document Attributes** dialog window allows the following settings:

Author — In this field you enter the name of the author as creator of the drawing.

Name of Attribute — In these four entry fields you can specify document attributes, that are displayed when you select the **Save as** command. The date of original creation must not be specified, since it will be saved automatically with each drawing.

Value of Attribute — In these fields you enter values for the attributes you have defined in the left column, in case they don't change with each drawing. These values are optional, since they can be entered each time when a drawing is saved.

Post at 'Save as' — When you activate this option, the **Document Attributes** dialog will be displayed automatically when you select the **Save as** command.

Embed EMF/PIC — When you activate this option, a picture of the drawing will be saved automatically with the document each time you save a drawing. The **Height** and **Width** of this picture you can indicate in the respective entry fields. This preview picture will be displayed automatically in the **Open** dialog window and can be displayed in the ***Windows*** *Explorer*.

Cancel — Clicking the **Cancel** button closes the dialog window and ignores all entries made in the **Document Attributes** dialog box.

OK — When you click this button, the dialog window is closed and all settings are saved for the current drawing.

In order to display document preview and attributes in the ***Windows****-Explorer*, you must select the direc-toöry containing ***Draft-Board****-drawings* which are saved with preview and attributes. Depending on the *Windows* version the *Explorer* must be configured accordingly.

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These setting are only valid for the current ***DraftBoard*** session. When you want to save these settings permanently you have to select the **Save Preferences** command in the **Preferences** submenu in the **Layout** menu.

Defining Document Properties

1. Select the **Document Properties** command in the **Preferences** submenu of the **Layout** menu.

   The **Document Attributes** dialog window is displayed.

2. Specify a name for the author of the drawing in the **Author** entry field.

3. Enter into the **4 Name of Attribute** fields attributes such as *Drawing Number* etc. that should be saved with the drawing. It is not necessary to specify an attribute for the original creation date,

since it will be automatically saved with the document as soon as an entry is made in the **Document Attributes** dialog box.

4. Specify values for the defined attributes, if these values don't change with each drawing. If not, they can be entered each time when you select the **Save as** command and the option **Post at 'Save as'** is activated.
5. Activate the option **Post at 'Save as'**, if you want to display the **Document Attributes** dialog each time you select the **Save as** command.
6. Activate the option **Embed EMF/PIC**, if you want to save a small picture of the drawing as preview with the document, that will be displayed in the **Open** dialog window and optionally in the ***Windows*** *Explorer* (**Win98**, **WinSE**, **WinME**, **Win2000**, **WinXP**).
7. Enter a pixel value for the **Width** and **Height** of the graphic. The default value is **600** x **600**.
8. Click the **OK** button.

   The **Document Attributes** dialog window is closed and the specified values are saved for the current drawing.

   If you want to save these attributes permanently as default values, you have to select the **Save Preferences** command in **Preferences** submenu in the **Layout** menu.

Displaying Document Properties when opening a drawing

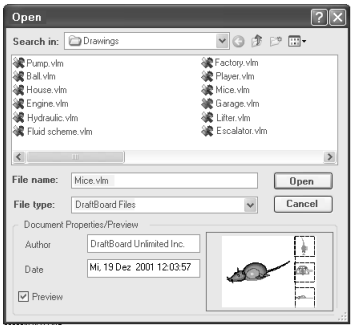
1. Select the **Document Properties** command in the **Preferences** submenu of the **Layout** menu.

   The **Document Attributes** dialog window is displayed.
2. Make all required entries.
3. Click the **OK** button.

   The **Document Attributes** dialog window is closed and the specified values are saved for the current drawing.
4. Save the drawing using the **Save as** command.
5. Close the drawing.
6. Select the **Open** command from the **File** menu.

   The **Open** dialog window is displayed.
7. Select the drawing that you just have saved.

   In the **Open** dialog window a *Preview* of the saved drawing (in case this option was activated for this document), the *Author* (if defined) and the *Original Creation date*. All other defined attributes for the drawing are stored in the **Document Attributes** dialog window after opening the drawing.



Displaying Document Properties in *DraftBoard*

1. Open a drawing that was saved with document properties you have specified in the **Document Attributes** dialog window.
2. Select the **Document Properties** command in the **Preferences** submenu of the **Layout** menu.

   The **Document Attributes** dialog window containing the attributes saved with the **Save as** command is displayed.

Displaying Document Properties in the Windows Explorer

To display *document properties* and *document previews* in the ***Windows****-Explorer*, you must go to the directory that contains ***DraftBoard*** *drawings*, that were saved with **Previews** and **Attributes** and activate the option **As Web Page** in the **Views** menu of the *Explorers*. ToolTips don't work with all Operating systems.

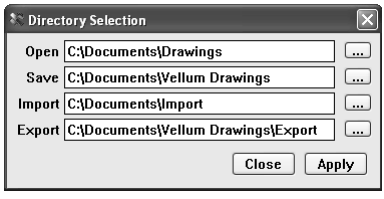
Default Directories

For *Opening*, *Saving*, *Im-* and *Exporting* documents you can set default directories. Default directories you can set with the **Directories** command in the **Preferences** submenu of the **Layout** menu.

Directories

This command sets default directories for *opening*, *saving*, *importing* and *exporting* documents.

When you choose this command the following dialog box is displayed:



After each entry field you find a button labelled with three dots. Clicking this button opens the **Searching Folder** dialog containing a directory tree with the content of your hard disk, where you can choose the proper directory.

Opening drawings

A detailed description how to open files you find also in the *User Manual* for the *Operating system* of your computer.

The **File** menu contains the **Open** and **New** command to open a drawing. With **New** you create a new drawing while the **Open** command opens an existing drawing.

New Ctrl+N [Macintosh ⌘N]

The name of the sheet is only displayed in the **Title Bar** if more than one sheet exists.

This command in the **File** menu creates a new ***DraftBoard*** document. The new document has no name (the title bar shows **Untitled 1**), and is set with the default options, such as *pen style* or *grid display*. If you open more than one new document, the subsequent documents are numbered sequentially until you name them by saving.

Open Ctrl+O [Macintosh ⌘O]

*Note: Use the **Import** command if you want to use drawings of other format types, such as **DXF** or **IGES**, or if you want to bring in **ASCII** text.*

The **Open** command in the **File** menu opens an existing document, that was created with ***DraftBoard***. This document can be created by current or earlier versions of ***DraftBoard***. The document appears in the drawing area maintaining the same settings as the last time you saved it. If you made any changes to the default settings, such as editing pen styles or hiding layers, those changes remain in effect.

The **Open** dialog box allows you to specify the document you want and lets you change directories or folders if necessary.

WIN

Opening a document WIN

1. Choose **Open** from the **File** menu.

   The **Open** dialog box appears.

2. Display in the **Look in** *pop-up* menu the appropriate *Folder* containing the document you want to open.
3. Select in the file type you want to open from the **File Type** *pop-up* menu.

   The list box displays all files of the selected type available in that folder.

The default setting is **DraftBoard files**.

4. Click the **File name** you want to open in the list box.

   If you double-click on the file name, the document is opened without clicking the **Open** button.
5. Click the **Open** button.

Opening a document MAC

1. Choose **Open** from the **File** menu.
   The **Open** dialog box appears.
2. Open the appropriate *Folder* containing the document you want to open.
3. Click the **File name** you want to open in the list box.
   If you double-click on the file name, the document is opened without clicking the **Open** button.
5. Click the **OK** button.

Recent File List

Another way to access files that have been opened recently is through the **Recent File List** that appears in the **File** menu after the **Exit** command. This list contains the names and paths of the most recent files that have been opened by ***DraftBoard***.

To open a file from the **Recent File List**, simply select the file from the **File** menu. If the file has been moved since it was last used and the path is no longer accurate, ***DraftBoard*** will ask you to locate the file by providing the standard **Open** dialog box.

The number of files contained in the **Recent File List** can be set for *Windows* in the **DraftBoard.ini** file and in the **Recent.lst** file for the *Macintosh*. For more information, consult the *Customization* section in the *Setting the Environment* chapter.

Saving a Drawing

You can save a drawing by choosing either **Save** or **Save As** from the **File** menu. The file is stored in the folder you specify.

Save Ctrl+S [Macintosh ⌘S]

This command in the **File** menu saves the current ***DraftBoard*** *document* to its original folder. If you want to save it to a different folder or with a different name, choose **Save As**.

If you choose the **Save** command and the document has not been saved previously, the **Save As** dialog box appears automatically, allowing you to name the document and specify the folder in which to save it.

If you have named and saved the document before, a brief message appears when you choose **Save**, and the program pauses while it updates the information.

***Note**: You should save frequently. Even though your drawing appears on the screen, it is not stored on the disk until you save it. Hours of work can be lost because of a power failure. It is also important to save before performing any intricate, multistep manoeuvre. In that way, if the result is not exactly what you had hoped, you can abandon the file by closing it without saving.*

Save As

This command in the **File** menu saves the current document. A dialog box appears so you can name the current document, give it a different name, or save it to a different folder.

Saving a document with a different name WIN

1. Choose the **Save As** command from the **File** menu.
   The **Save as** dialog box appears.
2. If necessary, display a different folder.
3. Type the name you want to use in the **File name** box.
4. Select the type of file you want to save the drawing as in the **Save as type** dialog box.
5. Either press **Return** or click **Save**.

The default setting is **DraftBoard file**.

Saving a document with a different name MAC

1. Choose the **Save As** command from the **File** menu.
   The **Save as** dialog box appears.
2. If necessary, display a different folder.
3. Type the name you want to use in the **File name** box.
4. Either press **Return** or click **Save**.

Making a backup

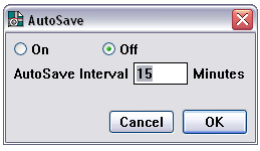
You should make a backup of your work in case you make many changes and want to go back to the original version. You can use the **Save As** command in the **File** menu and save the file with another name or use the **AutoSave** feature described in the next section.

It's also possible that your company makes periodic backups of your folders, and you can go back to a previous version that way.

Auto Save

This command in the **Preferences** submenu of the **Layout** menu directs ***DraftBoard*** to save a backup copy of your work periodically. If your computer hangs up for any reason, the work you did up to the last **Auto Save** will be recoverable.

Selecting the **AutoSave** command displays the following dialog box:



In the **AutoSave** dialog box you can *turn* **AutoSave** *On* or *Off* and specify the *time interval* (in minutes) between *AutoSaves*.

At the specified interval, ***DraftBoard*** creates an *AutoSave file* with the base of the original file name and a **.SAV** extension in the same directory or folder as the original file. **AutoSave** prompts users for a file name *before* creating an *AutoSave file* for untitled documents (created by using the **New** command from the **File** menu). If you supply a file name, this file name will be associated with the document and the *AutoSave file*. If you **Cancel**, **AutoSave** will skip that document, but the prompt will appear again at the next **AutoSave** interval. Files opened with the **READ ONLY** attribute are also *Auto Saved*.

Setting up AutoSave

1. Select **Auto Save** from the **Preferences** submenu of the **Layout** menu.

   The **Auto Save** dialog is displayed.

2. Switch AutoSave **ON** or **Off**.

   The default setting is **Off**.

3. Specify the time interval for *AutoSaving* in minutes.

   The interval can be from **1** to **60** minutes; the default setting is **15** minutes. An interval of **0** or less is equivalent to turning **Auto Save** off.

4. Click **OK**.

   If **AutoSave** is **ON** your work will be saved automatically for the time interval you specified.

If you want to save these settings permanently you have to **Save Preferences** in the **Preferences** submenu of the **Layout** menu.

Closing Documents

DraftBoard provides the **Close** and the **Quit** command in the **File** menu to close documents. If you want to close a drawing without ending ***DraftBoard*** you must use the **Close** command. Selecting the **Quit** command closes all open drawings and ends ***DraftBoard***.

Close Ctrl+F4 [Macintosh: ⌘ W]

This command in the **File** menu closes the current ***DraftBoard*** document (the one displayed in the top window). If other ***DraftBoard*** documents are open, they remain open when you close the current document. If you have made any changes since you last saved the current document, the **Save** dialog box allows you to save the changes. You can close the document with or without saving the changes.

You can also close the document by double clicking the **Control Menu** button (*Macintosh:* ***Close*** *box*) at the upper left corner of the title bar.

Quit Ctrl+Q [Macintosh: ⌘ Q]

This command in the **File** menu closes ***DraftBoard***. If you have made changes since you last saved, the **Save** dialog box allows you to save the changes. If you have more than one document open, an alert message allows you to save unsaved documents.

Data Exchange

DraftBoard like any other Computer program saves its data using its own native file format, that can be identified by the file extension **.vlm**. Even though all programs have native file formats that are not compatible to each other, it is possible to transfer data using neutral interchange formats such as **IGES** or **DXF** that standardize the data exchange. When you want to read a file of a non-compatible file format it is called ***Import****ing* files, when you want to save a file in a non-native format type it is called ***Export****ing* files.

It is also possible to import documents of native format type.

Limitations

Even if it is often advertised that programs can directly read and write file formats of competing products, you should be aware that importing files is always limited since the exchange is done through standardized file formats such as **DXF** and **IGES** that can offer only a compromise between the two file formats used.

These filters cannot transfer features specific to one application such as ***DraftBoard's*** *Smart walls* into another application where these features are unknown. Smart walls for example would be exported as a group of simple lines, all smart wall features known in ***DraftBoard*** would be lost.

Importing drawings

The import feature of ***DraftBoard*** allows you to use existing drawings from ***DraftBoard*** or other applications within a ***DraftBoard*** document.

Drawings created with *DraftBoard* on the *Macintosh* don't have to be imported but can be read using the **Open** command in the File menu.

The following file types can be imported

VLM ***DraftBoard*** native file format. Displayed option: *DraftBoard Files*. File name extension: ****.vlm***.

TXT **ASCII** text file. Displayed option: *Text Files;* File name extension: ****.txt***.

Spline This option allows to import *Spline coordinates*. ***DraftBoard*** creates then automatically a spline on the drawing area. File name extension: ****.txt***.

PICT The *Macintosh* graphics file format which uses object-oriented bitmaps or resolution independent graphics. *[Macintosh only]*

WMF **W**indows **M**etafile **F**ormat. An interchange format used by the *Windows clipboard* for graphics commands. Displayed option: *MetaFile;* File name extension: ****.wmf***. *[Windows only]*

BMP Microsoft Windows **Bitmap** file format. The file name extension **.bmp** is automatically appended to the file name. Displayed option: *Bitmap;* File name extension: ****.bmp*** *[Windows only]*.

EPS (AI) ***E****ncapsulated* ***PostScript*** *Format*, a data format for *PostScript* printers or for the import in *PostScript* compatible applications. ***DraftBoard*** supports the ***EPS*** *Format* of the ***Adobe*** *Illustrator* and compatible ***EPS*** *Formats*. Displayed option: *EPS/AI;* File name extension: ****.eps***.

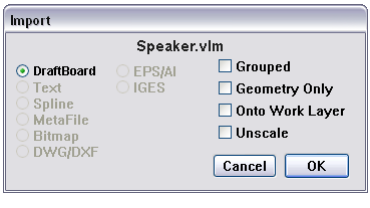
DXF/DWG AutoCAD ***D****ata* e***X****change* ***F****ormat* of ***AutoCAD***. Selecting this format in the **Import** dialog box and clicking the displayed **Options** button, you can choose from the ***AutoCAD*** *Formats* **BDXF**, **DXF**, **DWG** and *Versions* **10**, **11/12**, **13**, **14** and **2000/2002**. File name extension: ****.dwg*** or ****.dxf***.

Using the ***DWG/DXF*** *Import option* you can specify in a ***DXF*** *configuration file* additional import options such as *color handling*, *line patterns*, *line weights* or *text fonts*. More information about this configuration file you find in the *Appendix* of this manual.

IGS **IGES 4.0** ***I****nitial* ***G****raphics* ***E****xchange* ***S****pecification*, format to standardize the exchange of graphics. This file format allows as well to import *3D* ***NURB*** *Surfaces*. Displayed option: *IGES;* File name extension: ****.igs***.

Import

This command in the **File** menu imports a document from ***DraftBoard*** or another application into ***DraftBoard*** and places it in the current model. When you select the **Import** command the names of documents ***DraftBoard*** can read are displayed in a dialog box that works like the **Open** dialog box. When you select a document in this dialog box and click the **Open** button, ***DraftBoard*** detects the format of the document and displays the **Import** dialog box:



The graphic on the left shows the **Import** dialog box for *Windows*. In the *Macintosh* version you can't import **Bitmap** and **Meta Files** but **PICT** is supported.

In the **Import** dialog you can specify the following *Import options*.

You can ungroup the imported geometry with the **Ungroup** command in the **Arrange** menu.

Grouped	Imports the geometry grouped so it is a single unit in the ***DraftBoard*** document.
Geometry Only	Brings in only the geometry, leaving any *text*, *dimensions* and *crosshatching* behind.
Onto Work Layer	The imported geometry is placed on the work layer rather than the layer on which it was originally created.
Unscaled	The geometry is adjusted to correspond to the scale set by the **Drawing Size** command. If **Unscaled** is specified, the imported geometry is not scaled.

WIN

Importing a document WIN

1. Choose **Import** from the **File** menu.
   A dialog box appears similar to the **Open** dialog box.
2. Select the file type you want to import from the **Files of Type** *pop-up* menu.
   The list box shows all files of the selected type available in that directory.
3. Double-click the **File name** you want to import.
   The **Import** dialog box appears.
4. Specify any import options you want.
   If you have set a scale in drawing size, you should click the **Unscaled** option so that the geometry or drawing format you are bringing in is scaled the same as the geometry in the drawing.
5. Click **OK**.
   The file appears in the drawing area. If you want to save the file in its original format after editing, you must choose the **Export** command from the **File** menu.

MAC

Importing a document MAC

1. Choose **Import** from the **File** menu.
   The **Open** dialog box is displayed.
2. Select the file you want to import .
   The **Import** dialog box appears.
3. Specify any import options you want.
   If you have set a scale in drawing size, you should click the **Unscaled** option so that the geometry or drawing format you are bringing in is scaled the same as the geometry in the drawing.
4. Click **OK**.
   The file appears in the drawing area. If you want to save the file in its original format after editing, you must choose the **Export** command from the **File** menu.

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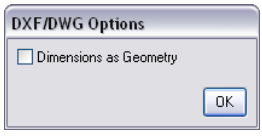
Importing DWG/DXF files

Important*: When you import a* ***DWG*** *or* ***DXF*** *file, the geometry is constructed according to the units set in* ***Preferences*** *when you choose the* ***Import*** *command. Be sure to set the appropriate units before you import* ***DWG*** *or* ***DXF*** *geometry.*

1. Choose **Import** from the **File** menu.
   A dialog box appears similar to the **Open** dialog box.
2. Select the file type **DXF** from the **Files of Type** *pop-up* menu.
3. Select the ***DXF*** *file* you want to import and click **OK**.
   The **Import** dialog box is displayed.
4. Select the option **DXF.**
5. Click **OK**.
   The **DXF/DWG Options** dialog box is displayed.



6. Specify the required options.

   When you choose the option **Dimension as geometry**, all dimensions are imported as geometry objects, if not they are imported as dimensions.

7. Click **OK**.

   DraftBoard begins creating the imported file on the drawing area.

Important Tips for importing DWG/DXF Files

! Important !

- When you import a ***DWG*** or ***DXF*** *file*, the geometry is constructed according to the units set in **Preferences** when you choose the **Import** command. Be sure to set the appropriate units used in the ***AutoCAD*** *File* before you import **DWG** or **DXF** geometry.
- When you import a ***DWG*** or ***DXF*** *file*, the **Drawing Scale** in the ***DraftBoard*** **Drawing size** dialog box in the **Layout** menu must be set to **1:1** to make sure that the imported drawing is correctly scaled.
- ***DraftBoard*** supports only **256** layers. If you have a ***DWG*** or ***DXF*** *file* containing more than 256 layers, all geometry on the additional layers will automatically be placed on the current work layer in ***DraftBoard***. If you don't want this to happen, reduce the number of layers to a maximum of **256** in the application you will be creating the ***DWG*** or ***DXF*** *file* in.
- When you get, after importing a ***DWG*** or ***DXF*** *File* into ***DraftBoard***, geometry not contained in the original ***AutoCAD*** *file*, you must purge any unnecessary blocks using the **PURGE** command in ***AutoCAD*** before exporting the file.
- If the file doesn't open correctly you should mark the options **Onto Work Layer** and **Dimensions as Geometry** in the **Import** dialog box and import the file again.

Importing ASCII text from another document

1. Choose **Import** from the **File** menu.

   A dialog box appears similar to the **Open** dialog box.

2. Select the file name you want to import.
3. Click **OK**.

   The **Import** dialog box is displayed.

4. If **Text** is not selected, mark the **Text** option.
5. Click the **OK** button.

   The text appears in a text box in the drawing.

Importing Splines

When you import a text file that contains the coordinates for a spline ***DraftBoard*** will create the spline according to the imported coordinates.

1. Select **Import** from the **File** menu.

   A dialog box appears similar to the **Open** dialog box.

2. Select a text file that contains the coordinates for the spline.

   The **Import** dialog box is displayed.

3. Specify the import option **Spline**.
4. Click the **OK** button.

   DraftBoard begins creating the Spline.

Creating a text file for importing a spline

1. Use a *text editor*, *word processor* or *spreadsheet* to create a text file.
2. Input **X**, **Y** and **Z** values for your spline coordinates.

   The text file should be tab or space separated. Each line ends with a return. Line feeds after each return should have no effect.

   The text file should conform to the following columnar format:

   1 1 0

   2 2 0

You are allowed to specify decimal coordinates as well:

1.33 1.1 0

2.4 2.5 3.5678

Be sure that you do a **Return** after the last coordinate, if not ***DraftBoard*** will not import the coordinates specified in the last line.

3. Save the file as **Text only** and import into ***DraftBoard*** to create the spline.

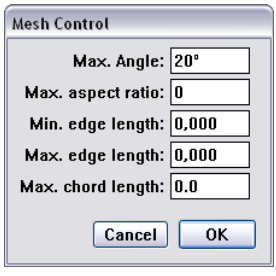
Exporting a Drawing

All settings for exporting a drawing are specified either automatically or by option buttons in the respective **Export** dialog windows displayed when exporting a file.

Only for the export of 3D Models general preferences can be set with the **Export 3D** command in the **Preferences** submenu in the **Layout** menu.

Export 3D

When you select this command the following dialog box is displayed:



All settings in this dialog window apply only for the *Export* of 3D Models, while identical settings in the **Visualization** dialog box apply for the *representation* of 3D Models.

The following settings in the **Export 3D** dialog box are valid for all Export data interfaces, that tessellate 3D Models for the export (**DXF/DWG**, **STL**, **SLP**) and apply only for triangulated surfaces.

Max. Angle This value describes the maximum possible *Exterior Angle* between two approximation triangles. Allowed are values **> 0°**. The default value is **20°**. The smaller the value the larger becomes the number of facets. This increases the precision of the model representation, but also increases the calculation time.

Max. Aspect Ratio This value describes the aspect ratio of the *min. edge length* to *the max edge length* of a approximation triangle. The value **1** equals an equilateral triangle. When **0°** is entered, this parameter is ignored. The default value is **0°**.

Min. Edge Length This value describes the minimum possible *Edge Length* of a approximation triangle. A small value increases the precision of the model representation, but also increases the calculation time. When **0°** is entered, this parameter is ignored. The default value is **0°**.

Max. Edge Length This value describes the maximum possible *Edge Length* of a approximation triangle. A small value increases the precision of the model representation, but also increases the calculation time. When **0°** is entered, this parameter is ignored. The default value is **0°**.

Max. Chord Length This value describes the maximum *Chord Length* between the ***NURB*** *Surface boundary* and the adjacent triangle edge of a approximation triangle. A small value increases the precision of the model representation, but also increases the calculation time. When **0°** is entered, this parameter is ignored. The default value is **0°**.

Export Data Interfaces

Occasionally, you may need to save a document in a format other than the ***DraftBoard*** format so you can use it with other application software. All settings for the export of drawings are specified either automatically or by option buttons in the respective **Export** dialog windows displayed when exporting a file.

Additional information about exporting data you find in the *Appendix* of this manual.

The following file types can be exported

DraftBoard 4.5 — ***DraftBoard's*** native file format.
File extension: ***.vlm**.

Vlm 3.0/4.0X — In ***DraftBoard Unlimited 4.5*** the file format was modified and is therefore incompatible to all previous versions. With the **Vlm 3.0/4.0X** format files can be exported into previous versions. File name extension: ***.vlm**.

Text — **ASCII** characters. This option is available only if in ***DraftBoard*** the **Text** tool is the active tool.
File name extension: ***.txt**.

BOM Attributes — This format exports all ***B**ills **o**f **M**aterial **A**ttributes* of a drawing as ***ASCII** Table*.
File name extension: ***.txt**.

Bitmap — *Microsoft Windows* **Bitmap** file, an interchange format specifically used for pixel graphics. File name extension: ***bmp**. *[Windows only]*

PICT — The *Macintosh* graphics file format which uses object-oriented bitmaps or resolution independent graphics *[Macintosh only]*.

MetaFile — The format is used by the *Windows Clipboard*, an interchange format specifically used for graphics commands. The precision equals the computer screen precision and exports graphics as vectors and pixel as represented on the screen. File name extension: ***.wmf*** for the 16 bit Window version or ***.emf*** for the 32 bit Window version. *[Windows only]*.

PNG — ***Portable N**etwork **G**raphics-Format*, a graphic format mainly used for distributing Internet graphics.
File name extension: ***.png**.

DXF/DWG — AutoCAD ***D**ata e**X**change **F**ormat* of ***AutoCAD***. This data interface supports the ***AutoCAD** Formats* **DXF**, **DWG** of *Versions* **10**, **11/12**, **13**, **14** and **2000/2002**. File name extension: ***.dwg*** or ***.dxf**.

For best possible translations see ***DXF** Configuration* in the *Appendix* of this manual.

EPS/AI — ***E**ncapsulated **PostScript** Format*, a data format for *PostScript* printers or for the export in *PostScript* compatible applications. ***DraftBoard*** supports the ***EPS** Format* of the ***Adobe** Illustrator* and compatible ***EPS** Formats*. File name extension: ***.eps**.

IGES — **IGES 4.0**. ***I**nitial **G**raphics **E**xchange **S**pecification*, format to standardize the exchange of graphics.
File name extension: ***igs**.

Stereo-Lithography — ***S**tereo-Lithography **T**essellation **L**anguage*, a format for *Rapid Prototyping*, that is required for the creation of prototypes. This format exports only three-dimensional structured surfaces and surfaced objects. File name extension: ***.stl**.

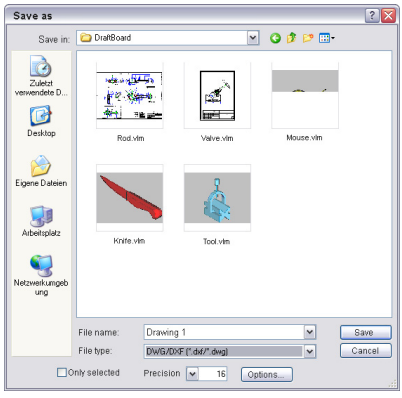
ProE/Render — *ProEngineer Render File-Format*, a format from **Pro/E** for Rendering 3D models. File name extension: ***.stp**.

Export

This command in the **File** menu saves a document in the format you specify.

Exporting a document

1. Choose **Export** from the **File** menu.
   The **Save as** dialog box appears.



2. Enter the desired file name.
3. Select the desired file type.
4. Set the required options.

   Depending on the selected export file type you can set several options, that are described in the next section of this chapter.
5. Click the **Save** button.

   The file is saved with the specified *file name*, *file type* and *export options*.

Export option for the available file types

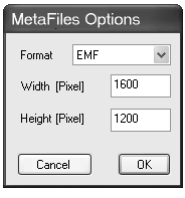
Depending on the selected export format you can set several options:

Only selected If you wish to export only selected objects on the screen, rather than the entire document, mark the option **Only Selected**.

Precision For some formats you can set the number of decimal places to specify the precision of the exported data.

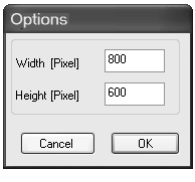
Options The following formats allow to display a dialog box by clicking the **Options** button in which you can set additional export options.

Metafiles The following export option are available:



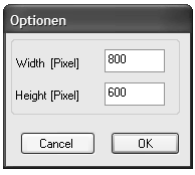
You can choose from the following Metafile Formats: **WMF** (16 bit Windows Version), **EMF** (**E**nhanced **M**etafile **F**ile: 32 bit Windows Version), **EMF** a Aldus specific ***EMF****-Format*. In addition you can specify the **Width** and **Height** of the graphic in Pixel units. These values specify the quality (resolution) of the graphic for the print out. The higher the resolution specified the better the print out quality.

PNG The following export option are available:



In this dialog window you can specify the **Width** and **Height** of the graphic in Pixel. These values specify the quality (resolution) of the graphic for the print out. The higher the resolution specified the better the printout quality.

Bitmap The following export option are available:

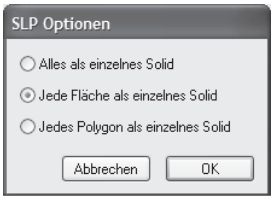


In this dialog window you can specify the **Width** and **Height** of the graphic in Pixel. These values specify the quality (resolution) of the graphic for the print out. The higher the resolution specified the better the print out quality.

STL The following export option are available:

The option **All triangles into one solid** doesn't exports the triangulized surfaces piece by piece, but assigns it to one solid.

SLP The following export option are available:



For the export you can choose from three options:

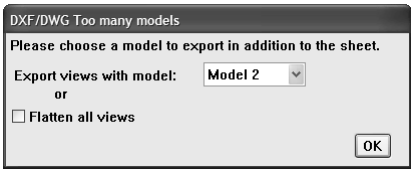
All into one Solid
Every face as one solid
Every polygon as one Solid

DWG/DXF The following export option are available:

In this dialog window you can select the ***AutoCAD*** *Formats* **DWG**, **DXF**, **DXB** and *Versions* **10**, **11/12**, **13**, **14** and **2000/2002**

In addition you can specify, if you want to export all hatches ungrouped.

Always when on the *current sheet* 2 or more models are displayed in *detail views*, the following dialog box is displayed:



In this dialog window you select either an additional model with its corresponding detail views or flatten all views to export them all.

Additional informations for exporting ***DWG/DXF*** *Data* you find in the Appendix of this manual.

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For ***AutoCAD*** *Version* ***12***, please choose *Version* ***11***.

If there are any problems with your exported *DXF files* you should use before creating any *DXF file* the **Remove Dots**, **Check Dimension** and **Select Duplicates** commands in the **Utility** menu.

If these commands are not listed in the **Utility** menu, they first must be added to this menu according to the *Customization* section in *chapter 4* of this manual. In addition you should read the text files *SelCopy.**txt***, *AntiDot.**txt*** and *Check-Dim.**txt*** in the **Function** folder in the **DraftBoard** directory.

Important Tips for the Export of DWG/DXF Files

- For exchanging data with other ***CAD*** *Programs* you should prefer the file format **DXF/DWG 2000** and for exchanging data with ***NC*** *Programs* the file format **DXF 12**.
- Until *version* ***13 AutoCAD*** doesn't accept any special characters such as a **&** character in layer names. Therefore remove all special characters from layer names before exporting any file into ***AutoCAD***. Otherwise ***AutoCAD*** will not import the file and generate an error message instead.
- In case line patterns are disorted (too big or too small) in ***AutoCAD*** after importing a ***DraftBoard DXF*** *File*, change the ***AutoCAD*** variable **LTSCALE** to a smaller number to display the line patterns at an appropriate scale.

Converting Files

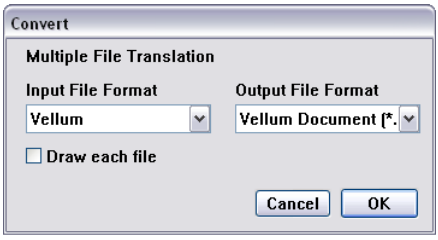
DraftBoard can not only import and export files but also convert files directly between different file formats.

Convert

This command in the **File** menu starts the **Convert batch translator** which allows you to *import* and *export* multiple files in an unattended mode. It imports all files of the same type from one folder and saves these files in the selected format into a destination folder.

The **Convert** command displays the following dialog box:

The **Convert** dialog box contains all file formats that can be converted by the *Windows* or *Macintosh* version of ***DraftBoard***. The graphic shows the **Convert** dialog box for *Windows*.



In the **Convert** dialog you can specify the following *Convert options*.

Draw each File If you mark this option you will see each file during the conversion process. If this option is not selected, ***DraftBoard*** will open a blank drawing window, perform the conversion, close the blank window and repeat the process for each file in the folder.

For large drawings or large quantities of drawings turning off **Draw each file** will speed up the operation.

Converting multiple files

1. Create two folders.
   One folder will be used to store the original files (the **FilesIn** folder), the other folder will store the converted files (the **FilesOut** folder).
2. Copy all files you want to convert into the **FilesIn** folder.
3. Select the **Convert** command from the **File** menu.
   The **Convert** dialog box appears.
4. Select the file type you are translating from in the **Input File Format** list.
5. Select the file type you are translating to in the **Output File Format** list.
6. Mark the option **Draw each file**, if wanted.
7. Click **OK**.
   The **Open file** dialog box displays.
8. Open the folder (**FilesIn**) that contains the files to convert.
9. Select a single file and click **Open.**
   The selected file must be of the same type as was selected for **Input File** format or ***DraftBoard*** will generate an error message.

10. Select a destination folder (**FilesOut**) for the translated files or create a new one.

    Specify a new extension to identify the converted files (it is reasonable to use the standard file type extension like **.dxf** for **DXF** files or **.igs** for **IGES** files.

DraftBoard will convert every file from the **Input** folder that is of the same type.

If you select a ***DraftBoard*** file in the **Input** folder, and requested convert to **DXF**, ***DraftBoard*** will convert every ***DraftBoard*** *file* in that folder to **DXF**, and place those **DXF** files in the **Output** folder you selected. If the folder also contains **IGES** files, they will not be translated.

CHAPTER

12

Parametrics

Using Parametrics

Parametric Drafting

Parametric Problems

Complex Parametric Drafting

Parametrics and Grouped Objects

Kinematic/Movies

Parametrics

DraftBoard's integrated parametrics feature allows you to create geometry without regard to its actual measurements. When you resolve geometry parametrically, you specify values for the dimensions and ***DraftBoard*** redraws the geometry to your specifications. The following are examples of parts well-suited to parametric definition:

- Containers that vary in size according to the needs of the product line.
- Hydraulic pistons that vary in size because of the duty loads.
- Valves that vary according to the diameter of the pipes to which they are attached.

In its simplest form, parametrics allows you to create a shape and then specify the exact measurements.

Of course, parametrics can be much more complex when you use variables, as you will see later in this chapter. While creating geometry is straightforward, when a part becomes complex, parametric dimensioning requires some skill and comprehension of how ***DraftBoard*** treats geometry.

For a deeper understanding of how *Parametrics* works we recommend therefore to study the *Parametric* chapter in the *Tutorial*.

Using Parametrics

The parametric mechanism is *point-driven*. It locates the points and then connects them with the appropriate geometry, much like the childhood activity of *connect-the-dots*. It is not unlike drawing with a compass, triangle, and T-square.

To begin, determine where the endpoints are and then draw lines to connect the points. Or you might figure out where the center of an arc should be, where the arc begins and ends, and use a compass to connect the points.

The **DraftBoard** folder contains a folder of symbols that use parametrics. You might retrieve a few of the symbols using the **Symbol** command in the **File** menu to become familiar with how parametrics works.

Basic Rules

1. **Define the geometry completely.**
   - Each object must relate to another object.
   - Dimensions must define every aspect of the geometry. (The geometry may be overdimensioned according to standard drafting practice.)
   - Do not include extraneous text as part of a dimension. (A single **#** is acceptable. The parametric mechanism recognizes **R #** to mean "a radius, measuring the real value of the geometry.")

Extraneous Text is explained in the *Dimensions* chapter.

2. **Keep it simple**.

   If the part is complex, construct small segments, resolving as you go, solving any parametric problem before proceeding.

3. **Return to the original part between tests**.

   Use the **Undo** command to revert to the original state of the parametric part between tests. In this way, you won't introduce an unwanted constraint.

For rules **c** and **f** each exist a special case. See as well under *Introduction* in the *Parametric* chapter of the *Tutorial*.

4. **Consider the following assumptions made by the parametric mechanism**:
   - **a** *Horizontal* and *vertical* lines maintain their orientation.
   - **b** *Connected lines* remain connected.
   - **c** * A construction point ***on*** the geometry of another object (not ***on*** a *construction point* of this object) remains connected to this geometry or its theoretical (mathematical) extension.*
   - **d** Lines *tangent* to arcs or circles remain tangent (if there is an ***endpoint*** at the ***tangency*** point).
   - **e** *Colinear lines* remain colinear if they overlap or share endpoints.
   - **f** * Each parametric construction must contain at least one *horizontal* or *vertical* line or alternatively a *horizontal* or vertical dimension.*

See as well *Assumptions Involving Tangency* in this chapter.
Lines are *colinear* if they have a common invisible *carrier line*. DraftBoard can recognize colinear lines only if they touch at least in one ***endpoint***.

5. **Be aware of the relationships that the parametric mechanism cannot recognize**:
   - Parallel lines may not remain parallel.
   - Perpendicular lines may not remain perpendicular.
   - Symmetrical geometry may not remain symmetrical.

Perpendicular lines remain only *perpendicular* if they have an *horizontal* or *vertical* orientation.

Using the parametric mechanism you create objects covering a wide range of validity due to their variable dimensions. This general validity expires whenever you assign true values to an object and transform it to a defined shape and size. Due to the rules of parametric this precise shape cannot be reverted to its original state under certain circumstances. Therefore return always to the original part between tests.

Creating Parametric Objects

When approaching the problem of resolving geometry, evaluate it as though you were going to draw it on paper. Remember the assumptions and requirements for geometric relationships as outlined in the previous *Basic Rules* section.

Parametric Dimensions

The essence of parametrics is in the dimensions. The dimensions may be the actual value (which you get by default), a constant which is not the actual value, a single variable, or an algebraic expression involving constants, variables, mathematical operators, functions, and conditional operators.

Constants are specific numeric values, such as specifying a radius as **.25** (inch). Variable expressions may be as simple as a single letter such as **L** (for **l**ength), or a mathematical expression such as **2 * Dia** (where **Dia** may be the diameter). The arithmetic operators are addition (**+**), subtraction (**-**), multiplication (*****), division (**/**), **mod** (the remainder after division—**%**), and exponentiation (******). See the *Appendix* for a list of functions and conditional operators.

However you choose to specify the dimensions, they must define all geometry and any relationships that exist between different parts of the geometry.

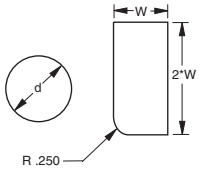
Creating parametric dimensions

1. Construct the geometry.
2. Display the dimension palette and choose the appropriate dimensioning tool.

   The status line displays a **#** symbol in the text box to show that the dimensions are entered as actual measurements.
3. Click the geometry to dimension as usual.

   The **Text** entry box in the status line is highlighted.
4. Type whatever expression, variable, or constant you want to use for the dimension, and press **Enter**. If you want the actual value to be used, do not change the **#** symbol in the text box. The expression you type replaces the **#** symbol.

Variables are case-sensitive: **D** is not the same as **d**.



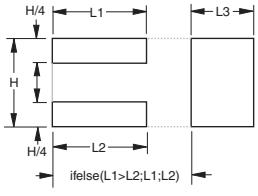
Example of an object with parametric dimensions.

Using conditional expressions in parametrics

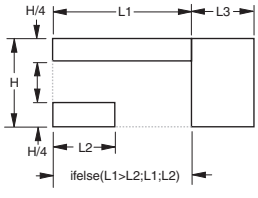
You can use the conditional operators as parametric dimensions. Create the dimension as usual and enter the conditional expression as text in the status line.

The example below shows a conditional **ifelse** expression.

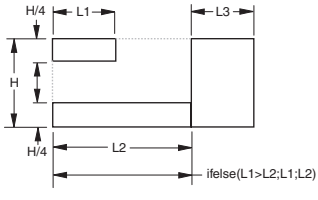
Using an **ifelse** condition the variable is set to the *first value* if the condition is **true** and it is set to the *second value* if the condition is **false**.



When the above **ifelse** statement is resolved, the rectangle with the larger value (**L1** or **L2**) will touch the **L3** rectangle.



The length represented by the **ifelse** expression equals **L1** because **L1** is greater than **L2**.



The **length** represented by the **ifelse** expression equals **L2** because **L2** is less than **L1**.

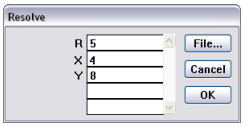
Resolving the Parameters

Once you have constructed the geometry and added parametric dimensions, use the **Resolve** command in the **Parametrics** submenu of the **Edit** menu to specify the values for the variables and redraw the geometry.

Resolve

This command in the **Parametrics** submenu of the **Edit** menu allows you to redraw geometry automatically to fit specified dimensions. This parametric feature enables you to draw a geometric shape without regard to measurements and then have ***DraftBoard*** redraw the same shape to the values you specify.

The **Resolve** command cannot resolve *ellipses* and *splines*, unless the *ellipse* or *spline* is contained in a group.



Using the Resolve command with parametrics

1. Create the geometry.
2. Dimension all geometry by using variables, constant values, and expressions. (You must dimension the essential, related geometry so ***DraftBoard*** can reconstruct the geometry. Note that ***DraftBoard*** cannot identify parallel or colinear lines.)
   - Select the appropriate dimensioning tool.
   - Click the geometry to be dimensioned.
   - Enter an expression (such as **1.5, x, x+3*y**) in the text box in the status line.
   - Press **Enter**.
3. Select the geometry and dimensions to be resolved.
4. Choose **Resolve** from in the **Parametrics** submenu of the **Edit** menu.
5. If necessary, enter the values you want to assign to the variables.
   You can enter mathematical expressions, fractions, and decimals in the entry boxes of the **Resolve** dialog box. You can use different units as long as you are specific, for example **2'6"**.
6. Optional step: If you want to anchor a point on the geometry at a particular location, click the point. The point remains in the same location after you resolve the geometry. An example appears later in this section.
7. Click the **OK** button.
   If the parametric dimensions defined the geometry properly, the geometry is redrawn as you specified. The dimensions remain as variable expressions.

If ***DraftBoard*** cannot resolve the parametrics, a message box displays information about the problem. The *Parametric Problems* section later in this chapter describes typical problems you may encounter. If you have not given all of the dimensions required to draw the geometry, the geometry is divided into unrelated groups. The missing information determines how the groups relate to one another.

An alert box shows how many groups exist. To see each group, click the **Next** button. Examine the groups shown to determine why the position of each one is not relative to any other. A group consisting of a single point is particularly telling. (This investigation can require some clever thought because the solution may not be obvious.)

Specifying Parametric Variables in a Text File

If you frequently use parametrics which contain many variables, you can create a text file to specify the variables and avoid entering them in the **Resolve** dialog box each time you resolve the parametrics.

Create a text file which alternates the variables and the values, using white space (**Spacebar**, **Tab**, or **Return**) between entries. ***DraftBoard*** assumes the first entry is a variable and the second is the value for that variable. Your text file might look like the examples below (or any other arrangements).

Height 12 R 17 Diameter 5

Height 12 R 17 Diameter 5

Height 12 R 17 Diameter 5

If you use a spreadsheet like Microsoft **Excel** for creating that text file you have to type in each cell the variable name **and** the value separated by a white space and export it as text file. If your text file contains more variables and values than you need to resolve the geometry, ***DraftBoard*** automatically imports only the needed variables and values.

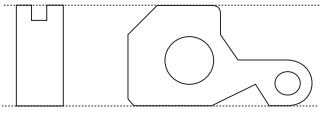
Using a text file for parametric variables

1. Create the text file using any word processor or spreadsheet and saving it as **Text only**.
2. In ***DraftBoard*** select the geometry and parametric dimensions.
3. Choose **Resolve** from in the **Parametrics** submenu of the **Edit** menu.
   The **Resolve** dialog box displays.

4. Click **File** to specify the values for the variables automatically.
5. Select the text file name.
   The variables and values are read from the file.
6. Specify an anchor point, if you want one.
7. Click **OK**.
   The geometry is resolved accordingly.

Anchoring the Geometry

You can anchor the geometry by clicking the anchor point when the **Resolve** dialog box appears. For example, in the *2D Tutorial*, you used parametrics to create the side view.



If you were using this side view as parametric geometry, you would want to anchor the upper-right corner so that it remains aligned with the front view after resolution.

Changing the Dimensions to Actual Measurements

Once you resolve parametric geometry, the dimension notations continue to show variables and constants. If you want the geometry to reflect the actual measurements, you have two options. You can edit the dimensions or save the geometry as a symbol. Follow these steps to change a resolved parametric dimension to a real value:

1. Choose **Selection Mask** from the **Edit** menu.
2. Highlight the four dimension types in the box on the left.
   Only dimensions are selectable.
3. Choose **Select All** from the **Edit** menu.
   All dimensions are selected.
4. Choose **Edit Objects** from the **Edit** menu.
5. Change the entry in the text box to a **#** symbol.
6. Click **Apply**.
7. Close the **Edit Objects** and **Selection Mask** dialog boxes.
8. Click the **Selection tool** to cancel the effect of the Selection Mask.

If you want different formats for different types of dimensions, select each type separately.

Parametric Drafting

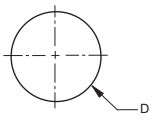
This section describes how to modify standard drafting practices to accommodate parametrics. It describes three examples with three topics to illustrate how to define parametric geometry completely.

Remember:

1. Define the geometry completely.
   - Dimension all geometry
   - Relate all geometry
   - Do not include extraneous text
2. Return to the original part between tests.

Dimension All Geometry

Centerlines require consideration if they are to be resolved properly.



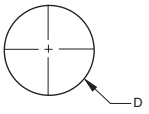
PROBLEM

If you create the circle shown on the left with two centerlines, parametrics can resolve the circle but not the *centerlines* because the ***endpoints*** of the lines are *not defined*.

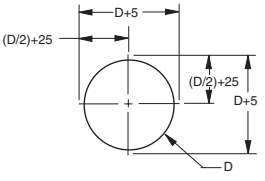
SOLUTIONS

If you look at this problem as a *connect-the-dots* problem, you can see that the *centerlines* have *no dots* to define them.

1. You can construct the *centerlines* so their endpoints are ***on*** the circle, as shown here.

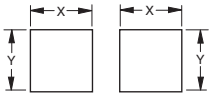


2. You can include *dimensions* for the *centerlines*. The dimensions should be variables based on the diameter of the circle so when you specify the diameter of the circle, the centerlines are drawn relative to the diameter.



Relate All Geometry

All parts of the geometry must be related.



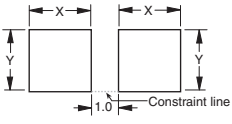
PROBLEM

Both squares could be drawn by the parametric mechanism, but there is no way to determine their *relative positions*.

SOLUTION

If you want different formats for different types of dimensions, select each type separately.

You can add a *constraint* line and *dimension* between the squares to connect the dots.



ADDITIONAL TIPS

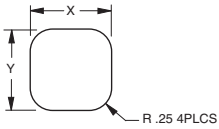
1. The added dimension is an example of using a **constant** value. Of course, you could have entered a variable expression such as **x+y** for the constraint line.

Each **dotted** line style (like the line style of the *Construction line*) will be hidden, when a symbol is placed or printed. Therefore you should use this line style for all invisible parts of an object.

2. **Constraint** lines are ordinary lines usually drawn with the **Single Line** tool. They can be any line style, but they should be different from the lines of the regular geometry. The **Construction** line style is a good choice.
3. If you place the **constraint line** and **dimension** on a layer (named **Constraints**), you can hide that layer when you plot the drawing.

No Extraneous Text

You must dimension all geometry because the parametric mechanism does not understand *text* and cannot detect *symmetry*.

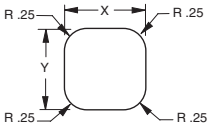


PROBLEM

If you dimension the radius of a filleted corner of a rectangle as **R .25 4 PLCS** to indicate four fillets, the parametric mechanism doesn't understand **4 PLCS** and doesn't know the corners are *symmetrical*.

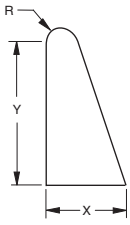
SOLUTION

Dimension each fillet.



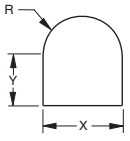
Return to the Original Part between Tests

If you resolve parametric geometry more than one time, you can introduce unintentional constraints.



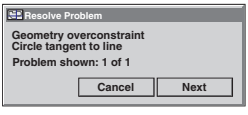
PROBLEM

If you first resolve this geometry by specifying that **R** is half the measurement of **X (R=X/2)**, the resolved geometry would appear as shown in the lower illustration.



You would now be able to resolve this geometry only with the same **X** value of **R * 2** because the undimensioned line is *vertical*, and according to the basic rules, *vertical lines remain vertical.*

If you tried to use a value for **R** or **X** other than the **R = X/2** relationship, the parametric mechanism would display this problem message:



SOLUTION

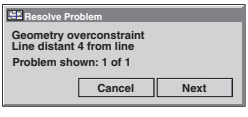
Use the **Undo** command to return to the original shape. You could also solve this problem by changing the geometry.

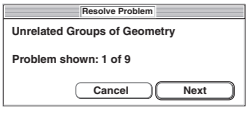
The **Undo** command is always available for any of the previous eight actions.

Parametric Problems

This section illustrates problems you may encounter when you resolve parametric geometry. It provides examples and shows you what caused the problem and how to solve it.

When you resolve parametric geometry, problems which prevent resolution appear in a message box, stating the type and number of instances of the problems.



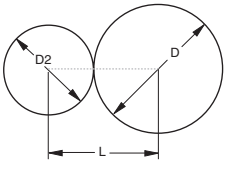


When a message appears, the problematic geometry appears as **thick lines** and **individual dots**. The lines indicate the geometry that the parametric mechanism can draw. **Dots** indicate unknown points. When you click the **Next** button, the next problem is shown. Examination of the relationships between the problems can help you discern what must be done.

Pay attention to **single points**! Remember, the parametric mechanism connects the dots. Frequently, one constraint or dimension can solve several problems at once.

Geometry Overconstrained

The basic rules say that you must define all geometry and relate every object to some other geometry within the parametric definition. When geometry is overconstrained, it is related in more than one way, so the geometry's specifications could be resolved to more than one answer.

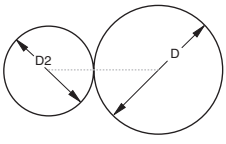


PROBLEM

The diameters of the circles indicate the relationship between the circles adequately because the *centerline* remains *horizontal* and the circles are *tangent* to each other. The **length** variable causes the problem because the length between the centers of the circles may not correspond to the sizes of the connected circles. **Note**: The *centerline* begins and ends at the centers of the circles.

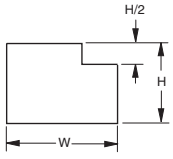
SOLUTION

Delete the **Length** dimension.



Unrelated Groups

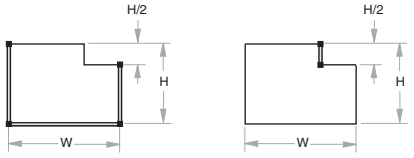
If you do not give enough dimensions, the parametric mechanism will find *two* or *more groups* of geometry, each of which is well-defined in itself, but *not related* to each other.

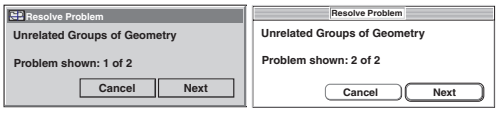


PROBLEM

A simple example of this problem is illustrated on the left, where the short *vertical line* is *not related* to the rest of the geometry. When you try to resolve this geometry, you get an **Unrelated group problem** message.

The first unrelated group appears in **bold** black lines. Click the **Next** button to highlight the second group.

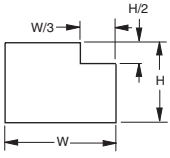




This problem is like the two squares shown earlier. Only one group could be drawn. The parametric mechanism does not know how to *relate* the *two* groups.

SOLUTION

You must decide how you want to *relate* the groups. Usually, you have many options to accomplish this. One alternative is to add the dimension **W/3**.

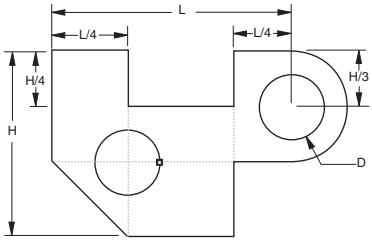


Unrelated Groups where One Group is a Single Point

A *single point* as an *unrelated* group is like a neon sign saying, *The problem is right here*!

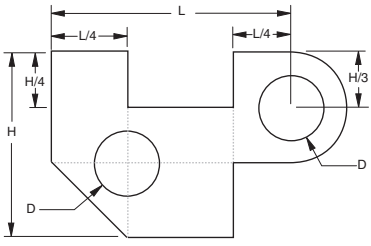
PROBLEM

The **Resolve problem** message may display a *single point*, such as the point on the left hole of this example. That point is the ***endpoint*** of the circle.

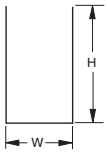


SOLUTION

On closer examination, you can see that the circle is *not* dimensioned. Therefore, the solution is to *dimension* the *circle*.



Another Unrelated Group



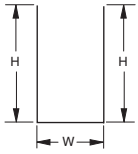
PROBLEM

In this illustration, the *height* of the right side of the part is dimensioned. You assume that the height dimension applies to the *right and left* side. The parametric mechanism does not make that assumption.

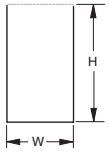
SOLUTIONS

There are three possible solutions to this problem:

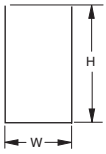
1. You can dimension the height on the *left*.



2. You can draw a *constraint line* to close the opening.

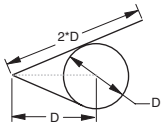


3. You can dimension the *height* from the *top* of the left line to the *bottom* of the right line. While this is not good drafting practice, it is very useful for parametric symbols.



Assumptions Involving Tangency

The parametric mechanism can identify tangent points but not the tangency of geometry.



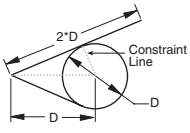
PROBLEM

The ***endpoints*** of the lower line that is ***tangent*** to and ends at the circle can be determined. However, the upper line is tangent to and extends beyond the circle and *cannot be defined* without more information.

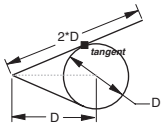
SOLUTIONS

Here are three possible solutions to the problem:

1. Draw a *constraint line* from the ***center*** of the circle to the *point of tangency* for the line that extends beyond the circle, as illustrated here.

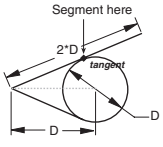


2. Use the **Rotate** tool on the **Transformation** subpalette to move the ***endpoint*** of the circle to the ***tangent*** point of the line that extends beyond the circle.



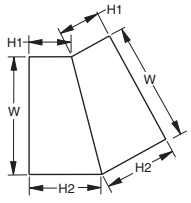
Pay attention to **single points**! Remember, the parametric mechanism connects the dots. Frequently, one constraint or dimension can solve several problems at once.

3. *Divide* the line into two segments which join at the ***tangent*** point.



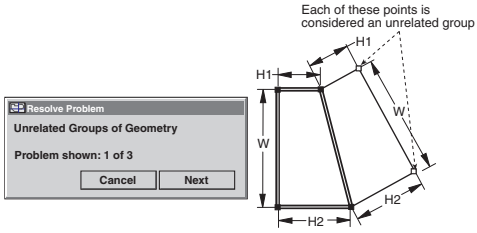
Relationships Involving Symmetry

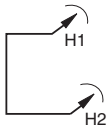
If you create geometry with a *mirrored* copy, the parametric mechanism cannot resolve and maintain the symmetry without more information.



PROBLEM

The problem message shows three *unrelated groups*: the trapezoid on the left and the two rightmost corners, as shown below.

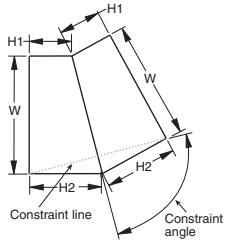




If parametrics swings an arc of the radius **H1** and **H2**, it doesn't know where the point is supposed to be on the arc. The point has a distance but *no direction* unless an angle dimension is added.

SOLUTION

Single points that are considered a *group* provide a clue to the solution. In this example, the parametric mechanism does not know where those points are, relative to the original trapezoid. You can solve the problem by adding a *constraint line* and *angle dimension* between that line and the mirror line.

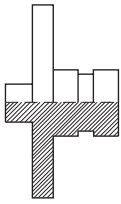


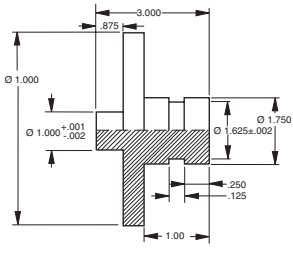
Complex Parametric Drafting

This section illustrates two complex examples of parametric drafting. These examples combine the information provided in this chapter so you can see how to use the parametric mechanism.

Example—Departures from Standard Drafting

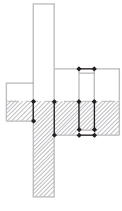
This example illustrates the parametric drafting practices that differ from standard drafting practices. The first illustration is the side view of an adapter without dimensions so you can see its shape clearly.





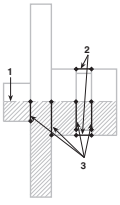
Here is the side view with typical dimensions added.

The illustration shows the side view with added constraint lines.

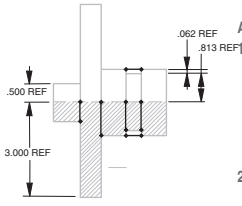


The next illustration shows the side view with numbered *constraint* lines.

1. The *centerline* does *not* extend beyond the geometry.
2. *Constraint lines* define the edges of the groove. Parametrics does not recognize *non-touching colinear lines*.
3. *Constraint lines* connect the ***endpoints*** of the upper and lower halves to relate the geometry. Parametrics does not recognize *non-touching colinear lines*.



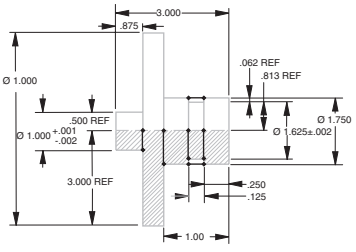
4. Four dimensions are added to relate the geometry to the centerline. Parametrics doesn't recognize non-touching colinear lines.



Additional Tips:

1. You should place the additional information on a separate layer so you can hide the layer when you plot the drawing.
2. If you use a yellow pen (which is hard to see against a white background) you won't confuse the additions with the actual geometry.

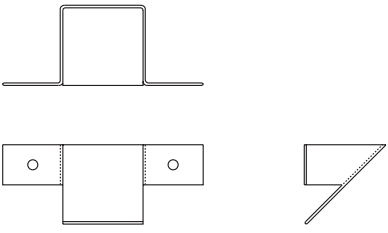
The illustration below shows the completed parametric drawing.



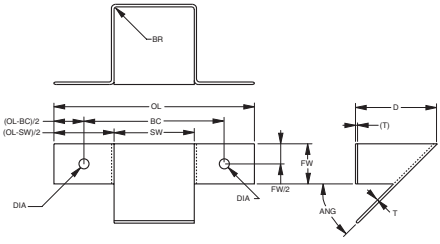
Example—Keep It Simple or Testing as You Go

When you have unconnected multiple views, the parametric mechanism can't determine the relationships between them.

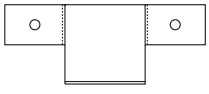
The coin chute example below describes the parametric drafting of three views.



You can dimension these views according to standard drafting practice, as shown.

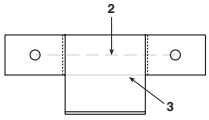


Section 1



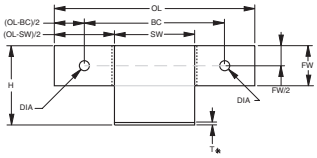
Parametric Drafting

1. Construct and resolve one section at a time. Solve any problems with this bracket before moving on.



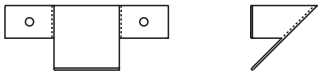
2. Add a *centerline* between the holes to maintain alignment.
3. Add a line connecting the lower edges of the flanges to keep the lines *colinear*.

The section below would resolve without problems, so you could go on to the next section.



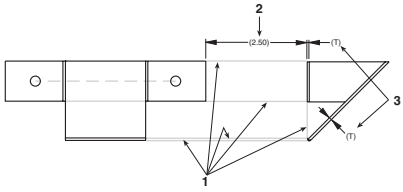
Once this view resolves properly, you can add the side view.

Section 2

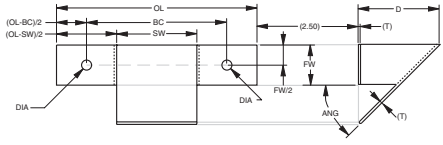


Parametric Drafting

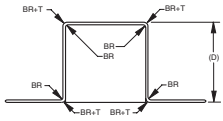
1. Add *constraint lines* to connect the geometry between sections.
2. Add a *dimension* to specify the distance between the sections. (See the two boxes problem earlier in this section.)



3. Add extra thickness (**T**) dimensions since all geometry must be defined.

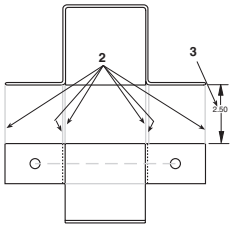


Section 3



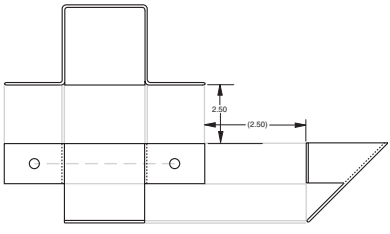
Parametric Drafting

1. Dimension each fillet, as explained earlier.

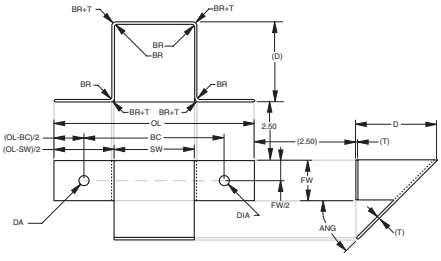


2. Relate this section to the rest of the geometry.
3. Add a dimension to specify the distance between the sections.

Three sections are related as shown below.



This is the completed parametric drawing.

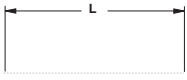


Parametrics and Grouped Objects

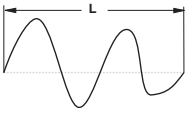
You can create many objects and then group them and treat them as a single parameterized object. There must be a framework on which the group sits. Dimension between two control points on the framework and then resolve the framework; the group changes accordingly, shrinking or expanding proportionately to the distance between the *control points*.

You can create a spline and group it.

Then create a framework line between the ***endpoints*** and dimension the line.



When you resolve for the length of the line (**L**), the spline changes accordingly.



Using parametrics and grouped geometry

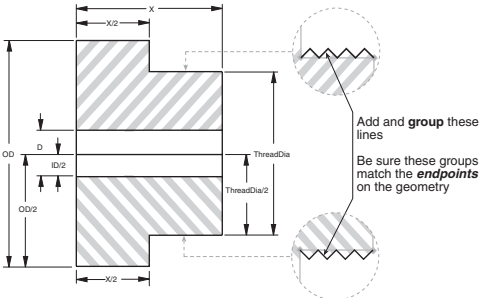
1. Create the geometry to be parameterized.
2. Select the geometry.
3. Choose **Group** from the **Arrange** menu.
4. Create a framework on which the grouped geometry sits.
5. Dimension between two *control points*. You can use only one variable per group.
6. Select the framework, the dimension, and the group.
7. Choose **Resolve** from the **Parametrics** submenu of the **Edit** menu.
8. Enter the value for the distance between the control points.
9. Click **O**K.

If you include a dimension as part of the group, the dimension will change when you resolve the group. It will not be used as part of the parametric solver.

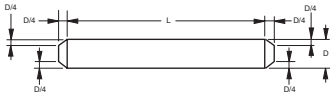
Rigid Links

You can group geometry into a rigid body and then attach the rigid body to parametric geometry in at least **two** points. When you resolve the parametric geometry, the rigid body will undergo the same geometric transformation.

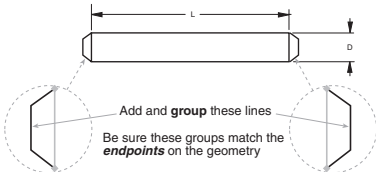
The following example shows a real-life example of using parametrics with grouped objects. Begin with a basic shape that resolves properly, as shown in the part below. Then add the thread groups. When you resolve the part, the threads also resolve.



In the next example, you can see the usual method of parametric drafting followed by an example showing how using parametrics with groups can simplify making changes to the part.

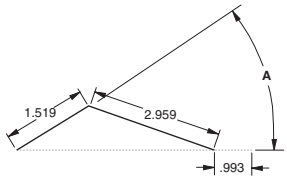


showing how using parametrics with groups can simplify making changes to the part.

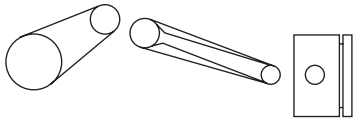


Triangle Link

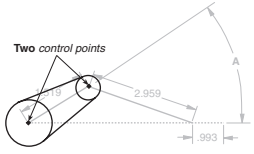
For a final example of using parametrics with groups, see how a triangle becomes the foundation for three different groups of a rod and piston. Begin with a parametric triangle that resolves with one variable, the angle **A**.

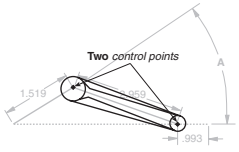


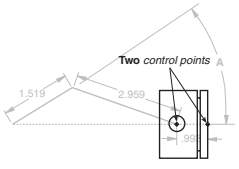
Then, create the geometry below and **group** it into three separate groups.

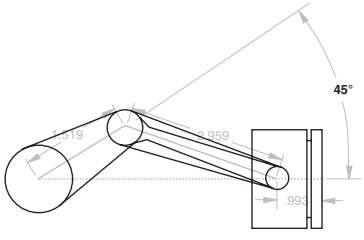


Once the groups are linked to the parametric geometry (**two** *control points* of the group match **two *endpoints*** of the line), you can **resolve** the triangle and the groups will change accordingly.

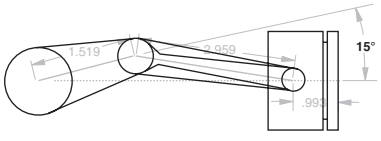








Resolved geometry where **A = 45°**



Resolved geometry where **A = 15°**

Kinematic

Kinematic is an enhancement to *Parametrics*: the animation of parametrized parts. To control the motion of these kinematic parts you can define specific parameters.

The **Calculate Motion** command in the **Parametrics** submenu creates and plays a series of frames which resolve the parametric geometry as specified.

If you define a *measure point* using the **Define Measure Points** command in the **Parametrics** submenu the *path* of this point can be tracked and displayed.

Creating a kinematic part

The following steps describe how to create a parametric part.

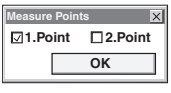
The **Kinematic** folder created by in the **Example** folder of the **DraftBoard** directory contains sample drawings for kinematic parts.
How you define a part related to an angle is described in the sections *Rigid Links* and *Triangle links* of this chapter.

1. Create a part having only one variable for the *Rotation Angle*.
   For *rotary motions* you should define an *angle* as the only variable. It is also possible to animate parts by resolving *longitudinal variables*.
2. Select the part including all dimensions using the **Selection** tool.
3. Select the **Resolve** command from the **Parametric** submenu in the **Edit** menu.
   The **Resolve** dialog box is displayed.
4. Test the part by entering different values for the variable **Angle** to be sure it resolves properly.
5. Save the part.

Defining Measure Points

Points whose path you want to track during the simulation must be defined as *Measure Points*. Any *Start-* and *Endpoint* of *Parametric dimensions* can be used as a *Measure Point*.

When you select a parametric dimension and choose the **Define Measure Points** command from the **Parametric** submenu in the **Edit** menu the following dialog box is displayed:



In this dialog box you can define the *Startpoint* (**1. Point**) and /or the *Endpoint* (**2. Point**) as *Measure Points*.

When you click **OK** the points are marked by a green cross in the drawing as *Measure Points*. When you animate the part the path of these points is tracked and displayed as a curve on the screen.

Defining Measure Points

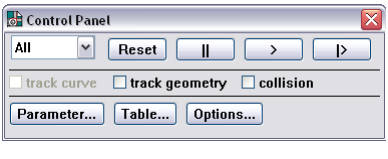
1. Select a parametric dimension.
2. Select the **Define Measure Points** command from the **Parametric** submenu in the **Edit** menu.
   The **Define Measure Points** dialog box is displayed.
3. Mark one or both endpoints of the selected dimension as *Measure Points*.
   The selected points are marked by a green cross.
4. Click **OK**.

Calculate Motion WIN

WIN

This command in the **Parametric** submenu in the **Edit** menu allows you calculate the motion of parametric objects.

When you select a parametric part including its dimensions and choose the **Calculate Motion** command the following dialog box is displayed:



The differences to the *Macintosh* **Control Panel** are described in the next section.

The kinematic **Control Panel** allows the following settings:

All — This list box contains all variables you have defined for the selected object. If you select **All** the motion will be calculated for all defined variables, if you select a single variable the motion will be calculated only for the selected variable.

II — The **Pause** button allows you to stop a simulation that was started with the **Play** button. To stop a simulation you must keep the **Pause** button pressed until the simulation stops.

> — The **Play** button starts a complete motion cycle and can only be stopped with the **Pause** button. When clicking the **Play** button the complete motion will be calculated from the *starting* value to the *ending* value, specified in the **Parameter** dialog box. When the simulation ends all variables are reset to their original values.

I> — The **Step** button starts a step-by-step simulation according to the value you have set in the **Increments** field in the **Parameter** dialog box.

Reset — The **RESET** button resets all variables to their original values. This button is only active when a simulation was started with the **Step** button, since using the **Play** button resets all variables automatically.

The **Reset** button resets only the variables to their original values, not the graphic. To reset the graphic you must use the **Undo** command in the **Edit** menu.

Track Curve — When this option is marked the path of all defined *Measure Points* is tracked and recorded during a simulation.

Track Geometry — When this option is marked the geometry for each step is recorded and displayed on the screen.

Collision — This options records all collision points between the animated part and some stationary geometry on the screen.

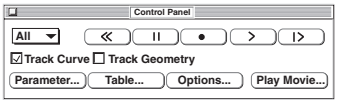
MAC

Calculate Motion [Movies] MAC

The *Macintosh* version of ***DraftBoard*** allows in addition to record and replay calculated motions as movies. Therefore the *Macintosh* **Control Panel** contains different buttons and options than the *Windows* version.

When you select a parametric part including its dimensions and choose the **Calculate Motion** command the following dialog box is displayed:

Collision Points cannot be calculated on the *Macintosh*.



The kinematic **Control Panel** allows the following settings:

<< — The **Rewind** button resets all variables to their original values. This button is only active when a simulation was started with the **Step** button, since using the **Play** button resets all variables automatically.

The **Rewind** button corresponds the **RESET**-button under *Windows*. The **Rewind** button resets only the variables to their original values, not the graphic. To reset the graphic you have to use the **Undo** command in the **Edit** menu.

II — The **Pause** button allows you to stop a simulation that was started with the **Play** button. To stop a simulation you must keep the **Pause** button pressed until the simulation stops.

> — The **Play** button starts a complete motion cycle and can only be stopped with the **Pause** button. When clicking the **Play** button the complete motion will be calculated from the *starting* value to the *ending* value, that you have specified in the **Parameter** dialog box. When the simulation ends all variables are reset to their original values.

● — The **Record** button starts a simulation and records all calculated steps as a movie. When you click this button the **Save as** dialog box is displayed where you can enter a name for the movie.

I> — The **Step** button starts a step-by-step simulation according to the value you have set in the **Increments** field in the **Parameter** dialog box.

Track Curve When this option is marked the path of all defined *Measure Points* is tracked and recorded during a simulation.

Track Geometry When this option is marked the geometry for each step is recorded and displayed on the screen.

Play Movie This option allows you to play a recorded kinematic motion as a movie in a separate window. When you select the option **Play Movies** the **File open** dialog box is displayed where you can select the movie.

When you record a kinematic motion as a movie using the **Record** button in the kinematic **Control Panel** a series of frames is created with different values for the variables. When these frames are put together they create a motion picture called *Movies*. The file is in **PICT** format and can
be imported into other applications.

Recording a movie MAC

1. Create a part that you would like to animate including its parametric dimensions.
2. Select the complete part including its parametric dimensions.
3. Choose the **Calculate motion** command in the **Parametric** submenu from the **Edit** menu.
   The kinematic **Control Panel** is displayed.
4. Specify all parameters and options in the **Parameter** and **Options** dialog boxes.
5. Click the **Record** button.
   The **Save as** dialog box is displayed.
6. Enter a name for the movie in the **Name** field.
7. Click **OK**.
   The kinematic motion is recorded and saved as a movie under the name you specified.

Playing a movie MAC

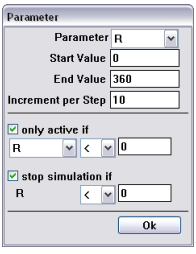
1. Choose the **Calculate motion** command in the **Parametric** submenu from the **Edit** menu.
   The kinematic **Control Panel** is displayed.
2. Click the **Play Movie** button.
   The **File open** dialog box is displayed
3. Select the **name** of the movie and click **OK**.
   A new window display the movie.
4. Use the VCR-like controls at the bottom of the window to replay the movie.

5. Click the **Close** box on the window when you are finished.

- End of MAC description -

Parameter

When you click the **Parameter** button the following dialog box is displayed:



The **Parameter** dialog box allows the following settings:

Parameter In this list field you select variables for which you want to specify values for the simulation.

*Important: To avoid possible parametric constraints during a simulation you should enter the **starting** and **ending value** as decimal numbers like **0,1°** to **45,1°** with an increment per step of **5°** for an angular variable.*

Start Value In this entry field you specify a *starting value* for the selected variable.

End Value In this entry field you specify an *ending value* for the selected variable.

In this entry field you enter a *step value* for the selected variable. This value defines at which increment per step the simulation should be calculated. The difference between the *starting* and the *ending value* should be divisible by the *step value*.

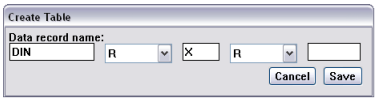
only active if When you mark this option you can specify a limiting value for the selected variable that defines at which value the variable will be included into the calculation.

When you mark this option you can specify a limiting value for the selected variable that defines at which value the simulation will be stopped.

Table

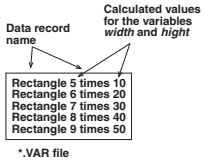
This command allows you to write the calculated values for each step into a table.

When you click the **Table** button the following dialog box is displayed:



In the **Table** dialog box you can enter a name for each calculated step (data record).

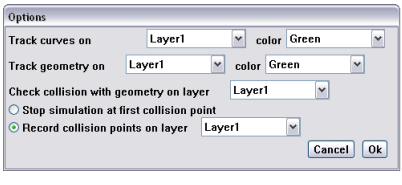
Each record contains the specified names and selected variables. If you simulate for example a rectangle with the values shown in the graphic above and **5** *Increments per Step* you will get a table with the following records:


Data record name
Calculated values for the variables width and hight
Rectangle 5 times 10
Rectangle 6 times 20
Rectangle 7 times 30
Rectangle 8 times 40
Rectangle 9 times 50
*.VAR file

The Table button creates a ***.VAR**-File* that you can use to create table based symbol libraries (see as well under Symbols in this manual).

Options

When you click the **Option** button the following dialog box is displayed:



The **Options** dialog box allows the following settings:

Track curves on In these list fields you can select a *Layer* and a *Color* for the tracked curve.

Track geometry on In these list fields you can select a *Layer* and a *Color* for the tracked geometry.

Check collision with geometry on layer In this list field you select the layer that contains the geometry for the collision check.

Stop simulation at first collision point If you mark this option for the collision check the simulation is stopped at the first possible collision point.

Record collision points on layer If you mark this option for the collision check you can select a layer on which all collision points will be recorded.

Calculate Motion

1. Create a part that you would like to animate including its parametric dimensions. Use as parametric variables an angle or a variable that allows a kinematic calculation.
2. Select the complete part including its parametric dimensions with the **Selection** tool.
3. Test the part using the **Resolve** command in the **Parametric** submenu from the **Edit** menu to be sure it resolves properly.
4. Choose the **Calculate motion** command in the **Parametric** submenu from the **Edit** menu.

   The kinematic **Control Panel** is displayed.
5. Select a variable for which you want to calculate the motion. If you want to use all defined variables for the calculation choose **All**.
6. Mark the option **Track Curve on** if you want to record the path of a defined *Measure Point*.
7. Mark the option **Track Geometry** if you want to display the resolved geometry for each calculated step.
8. Click the **Parameter** button.

   The **Parameter** dialog box is displayed.
9. Specify for each variable a *starting* and an *ending value* and the *Increment per Step*.
10. Click **OK**.

    The **Parameter** dialog box will be closed.
11. Click the **Options** button.

    The **Options** dialog box is displayed.
12. Select a **Layer** and a **Color** for the *Tracked Curve*, the *Tracked Geometry* and the *Collision Points* in case you have marked any of these options in the **Control Panel**.
13. Click **OK**.

    The **Options** dialog box will be closed
14. Click the **PLAY** button.

    The motion of the selected part will be calculated and displayed on the screen.

    When the option **Tracked Curve** is active the path of all defined *Measure Points* will be recorded on the selected layer and color.

    When the option **Tracked Geometry** is active the geometry will be calculated for each step and displayed on the selected layer and color.

    If you start the animation with the **Step** button only one step is calculated and displayed.
15. Choose the **Undo** command in the **Edit** menu to reset the geometry on the screen. In case you started the animation with the **Play** button all variables are reset automatically to its original values.
16. Click the **Reset** button to reset all variables to its original values in case you started the animation with the **Step** button.

*Important: The **Track Curves on** option is only active when at least one **Measure Point** is defined.*

Collision Check Win WIN

For a collision check you must draw at least one additional line or curve that can cause a collision with the animated part.

The collision geometry must be on a separate layer than the part you will animate and must be selected for the collision check in the **Options** dialog box of the kinematic **Control Panel**.

Before you can start the collision check you must mark the option **Collision** in the kinematic **Control Panel**.

For the collision check you must select the complete part including its parametric dimensions but without the collision geometry before you click the **Play** button.

CHAPTER

13

Symbols

Creating Symbols

Placing Symbols

Symbol Libraries

Symbols

DraftBoard offers powerful functions for creating and handling symbols. All symbol commands are listed in the **Symbols** submenu of the **File** menu.

The following section describes how to *create* and *place symbols* and the configuration of *Symbol Libraries*.

Creating Symbols

In ***DraftBoard*** you can place any drawing as a symbol and open any symbol as drawing.

DraftBoard distinguishes between:

- *Simple Symbols,*
  which have fixed dimensions,
- *Parametric Symbols*,
  which have parametric dimensions and as an option underlying value tables,

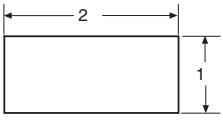
  and
- *Symbol Libraries*,
  that are configured with the **Libraries** command in the **Symbols** submenu, that can contain *simple* and *parametric symbols* and are handled exclusively with the **Symbolmanager**.

The following sections describe how to create the different type of symbols using a simple rectangle.

Creating simple symbols

Objects with fixed dimensions are called *simple* or *fixed* symbols.

1. Draw a rectangle with a *width* of **2 inch** and a *height* of **1 inch**.
2. Dimension the rectangle using the **Horizontal** and **Vertical dimension** tools.



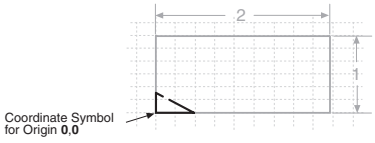
When you place a *fixed symbol* in a drawing using one of the **Symbol** commands, it is placed with its dimensions. These dimensions are fixed and can not be modified once the symbol is placed.

Defining the Insertion point of a Symbol

When you place a symbol in a drawing the *Insertion point* of the symbol is placed at that location you first clicked onto in the drawing area.

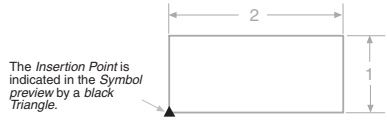
To define the *Insertion point* of a symbol you proceed as follows:

1. Create the object..
2. Place one *control point* at the origin (**0,0,0**) to specify the *Insertion Point* of the symbol.



A new document has its origin (**0,0,0**) in the center of the screen. To see the origin on the screen you must display the *Grid*. If you don't place a *Control Point* into the origin, the most left *Construction Point* is automatically used as *Insertion point*.

In the **Preview** window of all symbol dialog boxes the *Insertion point* of a symbol is shown as a small black triangle.

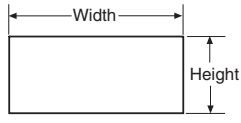


Creating parametric symbols

If you want to study the composition of a parametric symbol, you should open one of the parametric symbols shipping with ***DraftBoard*** in **Symbols** folder.

Objects with editable dimensions (using variables) are called *Parametric Symbols*.

1. Draw a rectangle at a any size.
2. Dimension the object according to the rules of parametric.
   In our example we overwrite the **#-Symbol** in the Status line with the variables **Width** and **Height**.



*Important: DraftBoard can extract the current values of parametric dimensions for **BOM** tables only when they start with a Capital letter. Therefore you should create only parametric variables that start with a Capital letter.*

3. Test the rectangle using the **Resolve** command in the **Parametrics** submenu of the **Edit** menu to be sure it resolves properly.
4. Save the symbol.

When you place a parametric symbol in a drawing it is placed without all dimensions that contain a parametric variable. The dimensions of parametric symbols can be modified after the symbol is placed in a drawing using the **Edit objects** command in the **Edit** menu.

Creating invisible Symbol parts

Each **dotted** line style (like the line style of the *Construction line*) will be hidden, when a symbol is placed or printed. Therefore you should use this line style for all invisible parts of an object.

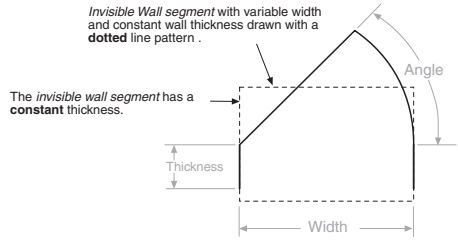
For all elements of an objects that should not appear with the symbol when it comes into a document, you must use pens with the line pattern **dotted**. For invisible dimensions you must modify the line pattern after placing the dimensions using the **Edit objects** command in the **Edit** menu, since dimensions always use a solid line pattern.

Creating smart symbols

The **Architect** folder of the **Symbols** folder contains smart symbols for doors and windows. Smart windows and doors are smart because they contain a smart wall segment. When you add one of these symbols to a smart wall, it breaks into the wall in the location you specify.

1. Choose a pen with the line pattern **dotted** in the **Pattern** submenu from the **Pen** menu.
2. Draw a smart wall thicker than the maximum thickness you expect to use and only as long as the final symbol will be.
3. Choose a pen style for the visible elements of the symbol.
4. Draw the visible parts of the symbol.

When you want to study a *Smart Symbol* open one of the *Door symbols* in the **Architect** folder of ***DraftBoard***.


Invisible Wall segment with variable width and constant wall thickness drawn with a **dotted** line pattern .
The *invisible wall segment* has a **constant** thickness.
Angle
Thickness
Width

5. Save the symbol.

Creating Value Tables for Parametric Symbols

When you place a parametric symbol in a drawing all parametric variables are listed in the **Symbols** dialog box. Instead of entering the final values into the related entry fields you can create underlying value tables that allow you to select predefined values for a selected symbol.

For that it is necessary to create a value table for each parametric symbol in a *text editor* or *spreadsheet*. Save the value table as an ***ASCII file*** under the name of the related symbol with the file extension **.txt**. (For the *Macintosh* you must create the file extension to **.txt** as well.) The file must be saved in the same folder as the related symbol.

Value tables may not only contain parametric variables but also user-defined attributes as *material* or *order number* that can be extracted in a *Bill of Materials*.

It is optional to specify values for the two required labels **PartID** and **Quantity** at the beginning of each table, since these attributes are automatically assigned to each symbol for a possible BOM extraction.

If you don't specify a value for the **Quantity**, the default value **1** is automatically assigned.

Labels for additional user-defined attributes as *Material* have to be defined after the labels for the defined variables.

Since these user defined attributes are not automatically attached to a symbol they must be assigned when creating the symbol using the **Symbol Attributes** tool.

Creating value tables

1. Create a *Norm part* at a scale of **1:1**.
2. Dimension the part using parametric variables.
3. Choose the **Attribute** command in the **BOM** submenu from the **Modules** menu.

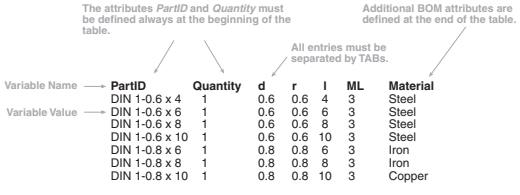
   The **Attribute** dialog box is displayed.
4. Define all attributes such as *material* that you would need for a *BOM*.

   Important: *Don't define the attributes **PartID** and **Quantity** since they are assigned to the symbol automatically with the underlying value table.*
5. Choose **Show Palette** in the **BOM** submenu.

   The **Attribute Tool Palette** is displayed.
6. Assign the defined attributes using the **Symbol Attribute** tool.

   How to use the **Symbol Attribute Tool** is described in the *Geometric Analysis* chapter in this manual.
7. Save the part under a appropriate name.
8. Note the names of all variables used for the part on a piece of paper.
9. Open a *text editor* such as *Notepad* or *SimpleText*.
10. In the first line, enter the two labels **PartID** and **Quantity** followed by all parametric variables you have noted for your norm part and finally all user-defined attributes you have assigned to the part.

    All entries must be separated by a **TAB** stop. At the end of the line, you must hit the **Enter** (*Windows*) or **Return** (*Macintosh*) key.
11. In the next line enter the respective values for the norm part. All values have to be separated by a **TAB** stop as well.
12. Enter the values line by line for all norm parts pressing the **Enter** or the **Return** key after each line. Your table should look similar to the following graphic after all values entered.


The attributes PartID and Quantity must be defined always at the beginning of the table.
Additional BOM attributes are defined at the end of the table.
All entries must be separated by TABs.
Variable Name
Variable Value
PartID Quantity d r l ML Material
DIN 1-0.6 x 4 1 0.6 0.6 4 3 Steel
DIN 1-0.6 x 6 1 0.6 0.6 6 3 Steel
DIN 1-0.6 x 8 1 0.6 0.6 8 3 Steel
DIN 1-0.6 x 10 1 0.6 0.6 10 3 Steel
DIN 1-0.8 x 6 1 0.8 0.8 6 3 Iron
DIN 1-0.8 x 8 1 0.8 0.8 8 3 Iron
DIN 1-0.8 x 10 1 0.8 0.8 10 3 Copper

In your value tables it is allowed to use units such as *mm*, *cm*, *Inch* etc. to have the correct dimensions for your symbols independent from the units set for the ***DraftBoard*** document.

13. Save the table as an ***ASCII*** *file* with the file extension **.txt** under the name and in the same folder you have saved the norm part.

When you now place a symbol with an underlying value table using the **Insert** command or the **Symbolmanager** all predefined norm parts from your newly created part family are listed and can be selected.

Important: *It can take a while when you open a symbol with an underlying value table for the first time since **DraftBoard** has to convert the ASCII table in an own format with the file extension **.var**.*

Editing value tables

1. Open the value table with the file extension **.txt** in the original *Text Editor* or *Spreadsheet* you used to create the table.
2. Modify the table.
3. Save the table as ***ASCII*** *file* (text only and carriage returns) under its original name.
4. ***Important:*** *Delete the file with the same name that has the file extension **.var**. It is in the same folder as the norm part with the file extension **.vlm** and the value table with the file extension **.txt**.*

Placing Symbols

The submenu **Symbols** in the **File** menu contains the two commands for placing symbols: **Insert** and **Symbolmanager**.

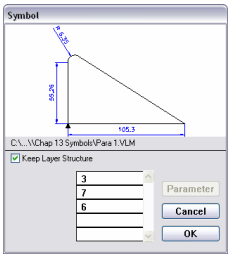
The **Insert** command allows to place any ***DraftBoard*** drawing and any norm part as a symbol.

Whereas the **Symbolmanager** command allows only to place and edit symbols from *Symbol libraries* that were configured with the **Libraries** command or imported as *Symbol libraries*. (See as well under *Symbol libraries* at the end of this section.)

Insert

Choosing this command from the **Symbols** submenu in the **File** menu displays the following dialog box:

If you want to enlarge part of the displayed symbol, bring the mouse pointer over the area of interest and press the left mouse button. As long as you keep the mouse button pressed the clicked area will be magnified.



The **Symbols** dialog box allows the following settings:

Preview window	The **Preview** window shows a picture of the selected symbol including all its dimensions.
Path	Below the **Preview** window you find the path to the folder where the selected symbol is stored.
Variables	For parametric symbols this list contains entry fields for all variables defined for the symbol.
Parameter	When you click the **Parameter** button the **Parameter** dialog box is displayed containing all attributes defined in the corresponding value table. If no value table exist the **Parameter** button is deactivated.

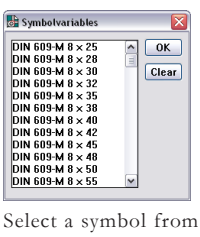
When you want to indicate a layer in the **Symbols** dialog box where the symbol should go you must define the attribute **Layer=** for the symbol using the **Define Attributes**.

To place the symbol in your drawing click or ***drag*** a vector on the drawing area. If you click, the symbol is inserted in the location you click, in its original orientation. If you ***drag***, the starting point of the ***dragged*** vector specifies the *Insertion point* for the symbol and the direction of the ***dragged*** vector indicates the *orientation*.

When you click **OK** the symbol is placed at the current scale of the drawing.

Placing a symbol

1. Choose the **Insert** command from the **Symbols** submenu.
   The **File open** dialog box is displayed.
2. Select the symbol file you want to use and click the **Open** button.
3. Enter a value for each parametric dimension or click the **Parameter** button, if an underlying value table for the symbol exists.
   The **Parameter** dialog box is displayed.


Symbolvariables
DIN 609-M 8 x 25
DIN 609-M 8 x 28
DIN 609-M 8 x 30
DIN 609-M 8 x 32
DIN 609-M 8 x 35
DIN 609-M 8 x 38
DIN 609-M 8 x 40
DIN 609-M 8 x 42
DIN 609-M 8 x 45
DIN 609-M 8 x 48
DIN 609-M 8 x 50
DIN 609-M 8 x 55
OK
Clear

The **Parameter** button is only active when a underlying value table exists.

4. Select a symbol from the **Parameter** list.
   The symbol values are transferred into the related entry fields of the **Symbol** dialog box.
5. Click the **OK** button.
   The **Parameter** dialog box is closed.
6. Edit the symbol values, if desired.

7. ***Drag*** a vector on the drawing area for the *insertion point* and the *Orientation* of the symbol.

   In the **Symbol** dialog box a *black triangle* indicates the *Insertion Point* of the symbol.

8. Click **OK**.

   The geometry resolves and appears in the current drawing at the location you clicked and sized as you specified. The inserted symbol is grouped and selected.

If no *Insertion Point* was specified when the symbol was created, the most left *construction point* of the symbol will be automatically used as insertion point.

Placing a symbol with its parametric dimension

If you want to bring a symbol including its parametric dimensions into the current document you must use the **Import** command in the **File** menu. The symbol is placed at its original size in the current document. If you select the symbol including its dimensions you can resolve it using the **Resolve** command from the **Parametrics** submenu in the **Edit** menu.

Standard, *non-parametric* dimensions are always placed with the symbol and must not be imported.

Editing parametric Symbols

If you want to edit parametric Symbols you must use the **Edit Objects** command in the **Edit** menu.

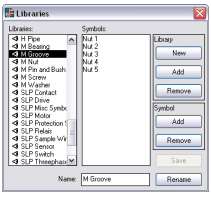
Symbol Libraries

In addition to the symbol libraries shipping with the ***DraftBoard*** you can create your own libraries using the **Libraries** command from the **Symbols** submenu in the **File** menu.

Libraries

This command allows to configure symbol libraries containing *simple* and *parametric* symbols.

The **Libraries** command displays the following dialog box:



The **Libraries** dialog box allows the following settings:

Libraries — This list field contains all available symbol libraries.

Symbols — This list field shows all symbols of the selected library.

New — This button creates a new library. When you click the **New** button the **Set Library file** dialog box is displayed where you can select a folder for the new *Library reference file* and overwrite the default library name **Library.vlb**. When you click the **Save** button the dialog box is closed and the library list shows the new library with the default name **Library 1** that you can modify in the Rename field.

Add — *Library*
This button allows to add an existing library. When you click the **Add** button the **Add Library** dialog box is displayed where you can select a *Library reference file* with the file extension **.vlb**. When you click the **Open** button the dialog box is closed and the selected library is added to the existing libraries.

Symbol
This button allows to add a symbol to the selected library. When you click the **Add** button the **Add Symbol** dialog box is displayed where you can select a symbol file with the file extension **.vlm**. When you click the **Open** button the dialog box is closed and the selected symbol is added to the existing library symbols.

Remove — This button removes a selected symbol or library from related list field.

Rename This button allows to modify the name of a selected symbol or library.

Save This button saves all changes made in the **Libraries** dialog box. When you close the dialog box using the **Close** field in the title bar, a message reminds you that all changes will be lost without saving.

Creating a new Symbol library

1. Choose the **Library** command from the **Symbols** submenu.
   The **Libraries** dialog box is displayed.
2. Click the **New** button.
   The **Set Libraries File** dialog box is displayed.
3. Overwrite the default name **Library.vlb** with an appropriate name.
4. Select a folder where you want to save the *Library reference file*.
5. Click **OK**.
   The dialog box is closed and the new library is added to the existing symbol libraries with the default name **Library 1**.
6. Select **Library 1**.
   Library 1 is displayed in the **Rename** field.
7. Overwrite **Library 1** with an appropriate name.
8. Click the **Rename** button.
   The new library name is displayed in the **Library list** field.

The name for the *Library reference file* must not be identical with the name you choose for the library in the **Library** dialog box. It is recommended to save the *Library reference file* in the DraftBoard folder.

Adding Symbols

1. Choose the **Library** command from the **Symbols** submenu.
   The **Libraries** dialog box is displayed.
2. Select the library you want add symbols to.
3. Click the **Add** button in the *Symbols* section.
   The **Add Symbols** dialog box is displayed.
4. Choose a *Symbol file* you want to assign to the selected library.
5. Click the **Open** button.
   The dialog box is closed and the new symbol is added to the **Symbol** list of the selected library.
6. Optional you may modify the file name using the **Rename** button.

Important: *When you assign a symbol to a library the symbol will not be copied but only the path to the folder where the symbol is stored. This path is saved in the* ***Library reference file*** *of the selected library. When you move a symbol to another folder the saved path becomes invalid and the symbol cannot be found anymore by the library.*

Adding Libraries

1. Choose the **Library** command from the **Symbols** submenu.
   The **Libraries** dialog box is displayed.
2. Click the **Add** button in the *Library* section.
   The **Add Library** dialog box is displayed.
3. Select a *Library Reference file* with the file extension **.vlb**.
4. Click the **Open** button.
   The dialog box is closed and the new library with all its symbols is added to the existing libraries.
5. Using the **Rename** button you can modify the file name of the new library.

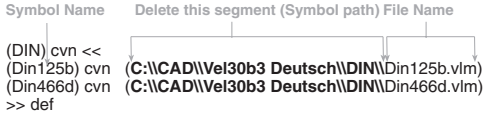
Making libraries accessible to other users

DraftBoard handles all symbol libraries in the reference file: *vsglib.fe* and all symbols of a library in the its related ***.vlb*** file.

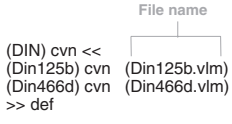
When you want to configure a library not only for your computer but for other computers as well you proceed as follows:

1. Create a new folder with the name of the future library.
2. Copy all symbols you want to assign to this library into the new folder.
3. Choose the **Library** command from the **Symbols** submenu.
   The **Libraries** dialog box is displayed.
4. Click the **New** button.
   The **Set Libraries File** dialog box is displayed.
5. Overwrite the default name **Library.vlb** with the name of the new folder containing the copied symbols.

6. Save the ***.vlb*** *file* in the newly created folder.
7. Click **OK**.
   The dialog box is closed and the new library is added to the existing symbol libraries with the default name **Library 1**.
8. Select **Library 1**.
   Library 1 is displayed in the **Rename** field.
9. Overwrite **Library 1** with the name of the newly created folder using the **Rename** button.
10. Click the **Rename** button.
    The new library name is displayed in the **Library list** field.
11. Select the new library.
12. Click the **Add** button in the *Symbols* section.
    The **Add Symbols** dialog box is displayed.
13. Select the new library folder.
14. Select the first copied *Symbol file*.
15. Click the **Open** button.
    The dialog box is closed and the new symbol is added to the **Symbol** list of the selected library.
16. Modify the file name of the new symbol if required using the **Rename** button.
17. Repeat steps **12** to **16** until you have assigned all symbols to the new library.
18. Click the **Save** button and close the **Library** dialog box.
19. Quit ***DraftBoard***.
20. Open the ***.vlb*** *file* in the new library folder using a *Text editor* or *Spreadsheet*.
    The file contains several *Control commands* and set in parentheses at the beginning of each line the *Symbol name* and at the end of each line the *path* of the symbol including its *file name*.



21. Delete in each line the *Symbol path*. Only the File name should remain in parentheses.



22. Save the file as ***ASCII*** *file* with the file extension **.vlb**.
23. Now you can distribute the newly created symbol folder including all symbols and the ***.vlb*** *file*.
24. When this folder is copied to another computer this library can be added using the **Add library** command.

The *Symbolmanager* checks when loading a library the path of the symbol saved in the corresponding ***.vlb*** *file*. If no path exists the *Symbolmanager* search the symbol in the folder of the ***.vlb*** *file*.

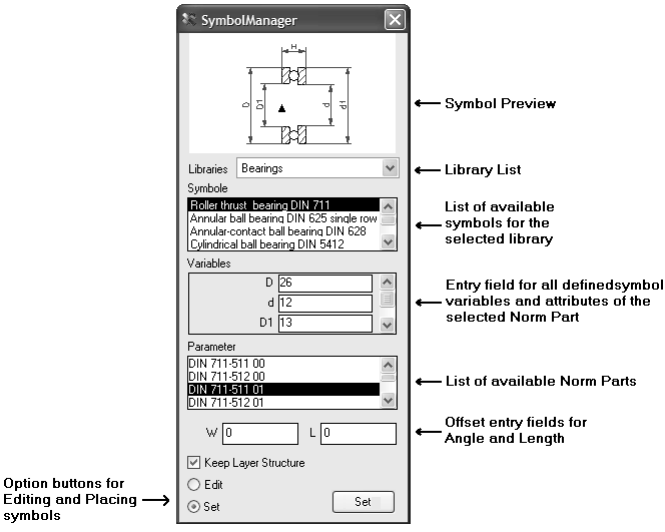
Handling of Symbol libraries

The *Symbolmanager* is the perfect tool for placing and editing symbols. Since the *Symbolmanager* handles exclusively symbol libraries it is recommended to group often used symbols into symbol libraries.

When the *Symbolmanager* is displayed you can place any number of symbols, replace and edit *Norm Parts* and offset the symbol *insertion point* by a specified distance and angle.

Symbolmanager

When you choose this command from the **Symbols** submenu in the **File** menu the **Symbolmanager** is displayed.



The **Symbolmanager** allows the following settings:

Preview window — The **Preview** window displays a picture of the selected symbol.

Library list — This list field displays all available libraries.

Symbols — This list field displays all symbols of the selected library.

Parameter — This list field shows all *Norm parts* of the selected symbol if an underlying value table exists.

Variables — This list field displays all *variables* of the *Norm part* selected in the **Parameter** list. All values of these variables can be modified.

A (Angle) — When you want to place a symbol offset to the first location you click on the drawing area you must specify in this entry field an **Angle** for the offset direction.

L (Length) — When you want to place a symbol offset to the first location you click on the drawing area you must specify in this entry field an offset **Distance**.

Keep Layer Structure — When you activate this option the original *Layer Structure* of a grouped symbol will be kept, when you ungroup it using the **Ungroup** command in the **Arrange** menu.

Edit — When you mark this option the **Set** button will show **Apply** and the mouse pointer changes its shape into a wrench tool. When you select a symbol on the drawing area all related data of the selected *Norm Part* are displayed in the *Symbolmanager*. You can modify the symbol data or select another *Norm Part*. When you click the **Apply** button the symbol will be reshaped or substituted.

Set — This option allows you to place symbols on the drawing area. To place a symbol you must ***drag*** a vector on the drawing area for the *insertion point* and the *orientation* of the symbol. When you click the **Set** button the symbol is placed according to the *dragged* vector.

When you have specified an offset angle and distance the symbol is placed offset to the insertion point according to the specified values.

Placing Library symbols

1. Display the *Symbolmanager* using the **Symbolmanager** command from the **Symbols** submenu in the **File** menu.
2. Select a library in the **Library** *pop-up* list field.
   The **Symbols** list field displays all symbols available in the selected library.
3. Select a symbol in the **Symbols** list field.
   If the symbol has an underlying value table all variations of this symbol are listed in the **Parameter** list field.
4. Select a *Norm Part* in the **Parameter** list field.
   All parametric *Variables* and *Attributes* of the selected *Norm Part* are displayed in the **Variables** list field.
5. If necessary you can modify the predefined values.
6. Mark the **Set** option.
7. If you want to place the symbol *offset* from the clicked *insertion point* specify an **Angle** and a **Distance** in the related entry fields.
8. ***Drag*** a vector on the drawing area for the *insertion point* and the *orientation* of the symbol.
9. Click the **Set** button.
   The symbol is placed according to the *dragged* vector.
10. Place additional symbols.
11. Close the *Symbolmanager* using the **Close** box in the upper right corner of the title bar.

When you want to indicate a layer in the **Symbols** dialog box where the symbol should go you must define the attribute **Layer=** for the symbol using the **Define Attributes**.

Editing Library Symbols

1. Display the *Symbolmanager* using the **Symbolmanager** command from the **Symbols** submenu in the **File** menu.
2. Mark the **Edit** option.
   The mouse pointer changes its shape to a wrench tool when you bring it over the drawing area.
3. Select the symbol on the drawing area you want to edit.
   All related data of the selected *Norm Part* including its symbol library are displayed in the *Symbolmanager*.
4. Modify the symbol data or select another *Norm Part* in the **Parameter** list field.
5. Click the **Apply** button. .
   The symbol will be reshaped or substituted. When you have specified any offset values the symbol is placed according these values on the drawing area.
6. Edit additional symbols.
7. Close the *Symbolmanager* using the **Close** box in the upper right corner of the title bar.

All symbol variables and attributes can be also modified using the **Edit objects** command.

***Note**: Depending on the symbol it can happen that after modifying a variable value it becomes impossible to resolve the symbol again.*

CHAPTER

14

Geometric Analyses

2D Analysis

Bill of Materials

Geometric Analyses

After you have constructed your geometry, you may want to do some analysis for either engineering purposes or for the production of your design part. ***DraftBoard*** includes a *2D Analysis* feature as well as a *Bill of Materials* feature. This chapter is divided into two sections to discuss these areas.

2D Analysis

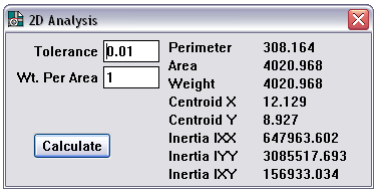
The **2D Analysis** provides sectional properties for the selected geometry, a feature that is useful for many calculations related to design and drafting.

2D Analysis

This command in the **Layout** menu displays the statistics on the selection—the length of the *perimeter*, the enclosed *area*, the center of gravity (centroid), and moments of *inertia*. The selection must be a closed figure.

It is also possible to determine the length of not enclosed objects such as *Splines* or *Arcs*. (See as well under *Perimeter* in this section of the manual.)

When you select the **2D Analysis** command the following dialog box is displayed:



The **2D Analysis** dialog box allows the following settings:

Tolerance A setting to determine the accuracy of calculations for curved objects. The smaller the tolerance is, the higher the accuracy will be. The default value is **0.01**. You can change the value by entering a new number.

Note that this *Tolerance* value has no effect on the precision of the resultant values. The precision of the result is set using the **Units** command under **Preferences** in the **Layout** menu.

The 2D Analysis mechanism analyses *arcs*, *circles*, *ellipses*, and *splines* as straight line segments that deviate from the true curve by no more than the stated tolerance value. The smaller the number, the greater the accuracy; however, the greater the accuracy, the longer the calculation time.

Weight Per Area A multiplier used to determine the actual weight of a part of constant thickness defined by a selected boundary. The Weight of the part is the product of the **Area** multiplied by the **Weight Per Area**.

The default entry is **1**. You can change the value by entering a new number. You can find the proper number for this entry in a handbook published by most material vendors, particularly for sheet steel and aluminium. Enter the weight for the thickness of the proposed material and click the **Calculate** button.

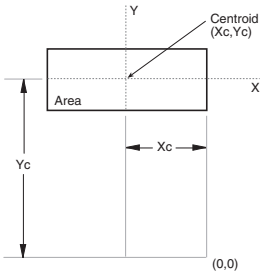
Perimeter The length of all segments which define the selected boundary. This is the only value that is calculated if the boundary of an object (such as a *Spline* or an Arc) is not closed.

For *circles*, *arcs*, and other curves, the accuracy is dependent upon the *tolerance setting*.

Area The surface area enclosed by the selected boundary.

For circles, arc, and other curves, the accuracy is dependent upon the tolerance setting.

Weight The product of the **Area** multiplied by the **Weight Per Area**.



Centroid X The **X** coordinate of the center of mass defined by the selected boundary.

Centroid Y The **Y** coordinate of the center of mass defined by the selected boundary.

Inertia IXX The moment of inertia about a central X-X axis which is parallel to the X-axis.

$$IXX = \int (y-Yc)2dA$$

Inertia IYY The moment of inertia about a central Y-Y axis which is parallel to the Y-axis.

$$IYY = \int (x-Xc)2dA$$

Inertia IXY The moment of inertia about a centroid.

$$IXY = \int (x-Xc)(y-Yc)dA$$

Performing 2D Analysis

The closed figure is an area that could be crosshatched. If you are in doubt about the area, simply crosshatch the geometry to see if it represents the area you want to analyse.
Once you crosshatch the area and you see that your selection is correct, you can choose **Undo** to remove the crosshatching and still maintain the selection.

1. Select the geometry that defines a closed boundary. You may want to use the **Tracer** tool to select the perimeter.
2. Choose the **Construction** command from the **Layout** menu.

   The **Construction** dialog box is displayed, the analysis is performed and the values are displayed in the dialog box.
3. If necessary, enter changes for the **Tolerance** and **Weight per Area** values.
4. Click the **Calculate** button.

   The analysis is recalculated and the new values are displayed in the dialog box.

Calculations

DraftBoard calculates the values for **2D Analysis** before the dialog box appears. If you make a change in the **Tolerance** or the **Weight Per Area**, you must click the **Calculate** button to recalculate the statistics.

The **2D Analysis** mechanism evaluates the boundary in the same way the crosshatching mechanism does; for example, ***DraftBoard*** considers a circle inside a bounded area to be a hole. Crosshatching does not fill the hole and **2D Analysis** does not include the area of the circle in the area calculation.

Displaying the centroid

1. Select the geometry defining the part.
2. Choose the **2D Analysis** command from the **Layout** manu.

   The **2D Analysis** dialog box is displayed.
3. Choose the **Construction** command from the **Layout** menu.

   The **Construction** dialog box is displayed.
4. In the **Angle** box in the Construction dialog box, enter **0;90**. (Be certain to separate the numbers with a semicolon.)
5. Enter the value for **Centroid X** displayed in the **2D Analysis** dialog box into the **X** box of the **Construction** dialog box.
6. Enter the value for **Centroid Y** displayed in the **2D Analysis** dialog box into the **Y** box of the **Construction** dialog box.
7. Press the **Enter** key.

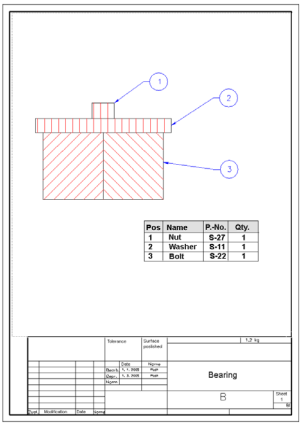
   The intersection of the two displayed construction lines show the centroid of the selected part.

Bill of Materials

Bill of Materials is an important extension to CAD programs for tracking and listing parts all the way through the product design and manufacturing process.

DraftBoard's *Bill of Materials* Extraction utility gives the product engineer the ability to attach characteristics (attributes) to the parts being designed, you can extract a Bill of Materials table from these parts. This Bill of Materials table can be printed separately or together with the drawing. In addition you can export the *Bill of Materials* to other applications for tracking of the informational data through the entire production process.

The *Bill of Materials* Extraction utility is completely integrated in ***DraftBoard*** and combines powerful functionality with an easy to use interface.



Generating a *Bill of Materials* is a three step process:

- defining attributes for object geometry in drawings
- assigning attributes to object geometry in drawings
- extracting user-defined and predefined attributes in the form of lists or ASCII-files for export purposes.

Attributes

Each object you create in ***DraftBoard*** has automatically assigned two different kinds of attributes:

Non-numerical attributes like the line *color* and the line *style*.

Numerical attributes like the *perimeter* and the *area* of a circle.

The ***Bill of Materials*** *utility* allows you to assign in addition

User-defined Attributes objects like a *Part Name* or the *Part Number* which you define and assign to object geometry after you have created the geometry.

With the *Bill of Materials* utility you can extract **numerical** and **user-defined Attributes**. *Non-numerical attributes* like line *color* and line *style* are not recognized by the *Bill of Materials*.

Defining Attributes

For defining attributes you choose the **Attributes** command from the **BOM** submenu in the **Modules** menu.

Attributes

This command allows you to *define*, *delete*, *redefine* and *activate* attributes. Active attributes (attributes which can be assigned) are indicated by a **checkmark** in front of the attribute name. Locked attributes (their values cannot be changed during assignment) are indicated by a **lock** icon in front of the checkmark.

The **Attribute** command displays the following dialog box:

The **Define Attributes** dialog box contains the following items:

New Clicking the **New** button creates a new attribute like **Attribute 1**, **2**, **3**... You can rename the attribute by typing the name in the **Attribute Name** entry box and clicking **Redefine**.

Delete Clicking the **Delete** button deletes the selected attribute from the attribute list.

Redefine Clicking the **Redefine** button does three things: First it renames a selected attribute with the name you have specified in the **Attribute Name** entry box; second, it assigns a value to the attribute which may be entered in the **Default Value** entry box; and third, it assigns the format for the attribute you selected from the **Format** pop-up list.

Normally you hide attributes only for defining ***BOM*** *layouts*. (See ***BOM*** *Layouts* later in this chapter). The section *Removing attributes* in this chapter describes how you remove attributes from object geometry.

Show/Hide With this button you show or hide attributes. Hidden attributes are not displayed in the Status Line upon assignment. Hidden attributes are indicated by a checkmark being removed in front of the attribute name.

Format In this pop-up menu you select a format for the attribute value. You can choose between three attribute formats: **Text**, **Number** and **Qty**.

Text **Text** is the default format and is the choice for all alphanumeric values (Text entries) like *Name*, *Material* etc.

Number **Number** is the choice for all values which are expressed by numbers which can be used for further calculations. A part number like **S3456** or **252-49** would be formatted as Text, since this value would never be used for any kind of calculation. If you use hyphens or dashes in your Part Number, you must use the **Text** format, not **Number**.

Qty An attribute would be formatted as **Qty.** if the *Bill of Material* should calculate the number of identical objects in a drawing. Normally this format only makes sense for an attribute which is named **Qty.** and has the default value **1**.

Attribute Name In this entry field you specify a general name for the attribute like *name*, *material, serial number* etc.

Default Value This entry assigns a specific value to the attribute. So you could assign the value of *steel* to the attribute of *material*. The value of an attribute can be changed in the Status Line before assigning it to object geometry or in the **Edit Objects** dialog box after it has been assigned.

Defining Attributes

1. Click **New** to create a new attribute.

   In the **Attribute Name** entry box the name **Attribute 1** is displayed.

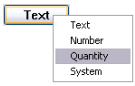
2. Rename the attribute by typing a name in the **Attribute Name** entry box, like COST.

3. Specify a value for the attribute in the **Default Value** entry box.

   The value you enter here can be changed in the Status Line during assignment, like 2.50. If you use the $ sign, the format must be **Text**, not **Number**.

4. Select a format for the new attribute. In the examples given above, the format is **Number**.

Click into the **Format** pop-up menu and ***drag*** the mouse pointer (with the left mouse button pressed) to the desired format. As soon as you release the mouse button the selected format displays in the ***pop-up*** *menu*.



5. Click the **Redefine** button.

   The new attribute name appears in the attributes list and the specified value and format is assigned to the attribute.

6. Close the **Define Attributes** dialog box by clicking the **Close** button in the title bar of the dialog box.

Defining Numerical Attributes

All attributes that are calculated and displayed in *2D Analysis* can be defined as *numerical attributes* for the *Bill of Materials*.

If you want to use these attributes in the *BOM*, you have to define them according to the following list in the **Define Attributes** dialog box:

- Perimeter
- Area
- Weight
- Centroid X
- Centroid Y
- Inertia IXX
- Inertia IYY
- Inertia IXY

Enter the names in the **Rename** field exactly as shown here. The bullet is not part of the name and should not be entered.

For *numerical attributes* enter only the attribute's *name*, since their *values* and *formats* are predefined.

Numerical attributes are not assigned but only defined for the extraction (see as well *BOM Layout* in this chapter). Therefore you must hide them before assigning any *non-numerical attributes*.

Therefore *Numerical attributes* are not displayed in the Status Line, since ***DraftBoard*** calculates the values automatically, as long as they are defined in the **Define Attributes** dialog box.

Defining parametrical Variables

It is also possible to define all *Parametrical Variables* that describe a symbol like *width*, *height* or *diameter* of an object as attributes for the *Bill of Materials*. *Parametrical Variables* are handled like *Numerical attributes*.

When defining these *parametric variables* you must enter them in the **Rename** field exactly as they are written in the related dimensions.

The *Bill of Materials* will extract then the current values of these *parametrical variables*.

Bill of Materials can extract the values of *parametrical variables* only when the variable's name begins with a Capital letter. Therefore should be the first letter of all parametrical variables you want to use in a BOM a Capital letter.

For *parametrical variables* you must specify only the attribute's *name*, since their *values* and *formats* are predefined.

Parametrical variables as a subgroup of *Numerical attributes* are not assigned but only defined for the extraction (see as well *BOM Layout* in this chapter). Therefore you must hide them before assigning any *non-numerical attributes*.

Defining Layers for Symbols

When you want to indicate a layer in the **Symbol** dialog box or in the **Symbolmanager** when placing a symbol you must define the attribute

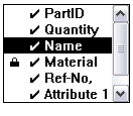
Layer=

If you specify a layer name in a *value table* or when inserting a symbol that doesn't exist it will be automatically created when inserting the symbol.

You must not specify the attribute's *value* and *format* since they are automatically assigned by ***DraftBoard***.

Locking and Unlocking Attributes

To lock an attribute simply click in front of the checkmark that is in front of the attribute's name.



The values of locked attributes cannot be changed during the assignment. In order to unlock an attribute you must click the lock icon off within the **Define Attributes** dialog box.

Changing the characteristics of an Attribute

1. Select the attribute in the **Define Attribute** dialog box whose *name*, *value* or *format* you want to change.
2. Type a new name or value, or choose a new attribute format.
3. Lock or unlock the attribute by clicking the lock icon **On** or **Off**.
4. Click **Redefine**.

   The new attribute name appears in the attributes list and the new value and format are assigned to the attribute.

Deleting Defined Attributes

1. Select the name of the attribute from the list box of the **Define Attribute** dialog box.
2. Click **Delete**.

   The selected attribute will be deleted.

Show or Hide Attributes

Numerical and *parametrical attributes* will not be assigned. You must them therefore hide before assigning any *non-numerical attributes*.

Only visible attributes (indicated by a checkmark beside the attribute name) can be seen during assignment. The user cannot edit hidden attributes.

1. Select the attribute you want to show/hide from the list box of the **Define Attribute** dialog box.
2. Click **Hide**.

   The checkmark besides the attribute name disappears.

Saving Attributes

With the **Redefine** button you rename an attribute or assign a new value or format to the attribute.

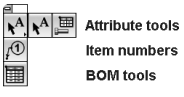
If you want to save all attribute definitions permanently you have to choose the **Save Preferences** command from the **Preferences** submenu in the **Layout** menu.

BOM Tool Palette

All *BOM tools* are on a separate palette which you can ***drag*** around the drawing area.

Show Palette/Hide Palette

This command from the BOM submenu in the Modules menu displays or hides the *BOM Palette*.



The B*OM Palette* contain all tools for assigning *User-defined* and *Symbol attributes*, for assigning *Item Numbers* and creating *BOM Tables*. You activate *BOM tools* like any other tool in the ***DraftBoard*** *tool palette*.

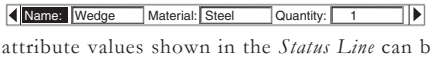
Assigning Attributes with the Attribute Tools

The first tool in the *BOM tool palette* contains the ***Attributes*** *subpalette* with the **Attribute Selection tool** and the **Symbol Attribute tool**.

If you want to assign attributes to an object which is made up of several individual objects—like a rectangle, which is made up of four single lines— you should group the object before assigning attributes. Otherwise, the attributes will be assigned to each individual object that is selected.

With the **attribute tools** you assign attributes to object geometry or mark an object as a symbol.

When you select one of these tools all attributes of the current ***BOM*** *layout* will be displayed in the *Status Line*. *Numerical*, *parametrical* and *hidden* attributes are not displayed in the *Status Line*, since their values will be not assigned but only extracted.



The attribute values shown in the *Status Line* can be entered or modified if they are not locked. If they are locked, the attribute has no entry box.

Attribute Selection Tool

With this tool you can assign attributes of the current layout to a selected object.

Assigning attributes

1. Choose the **Show Palette** command from the **BOM** submenu in the **Layout** menu.

   The ***BOM*** *Tool Palette* dialog box is displayed.

2. Select the **Attribute Selection** tool.
3. Select with the **Attribute Selection** tool an object you want assign attributes to.
4. Enter the desired values in the entry box in the *Status Line*.

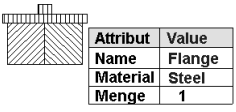
   Attribute values which are locked cannot be edited in the *Status Line*.
5. Press the **Enter** key (*Macintosh:* ***Return*** *key*).

   Important Note*: The attributes will be only assigned when you hit the* **Enter** key (*Macintosh:* ***Return*** *key*).

Before you can assign attributes they must be defined and a ***BOM*** *Layout* must be selected. For more information about the ***BOM*** *Layouts* see the *Bill of Materials Layouts* section later in this chapter.

Symbol Attribute Tool

With the **Symbol Attributes** tool you mark an object with a small table containing all assigned attributes.



Attribut	Value
Name	Flange
Material	Steel
Menge	1

When you insert this object with the **Insert Symbol** command or the **Symbolmanager** all attributes contained in this table will be listed in the **Symbol** dialog box. Values of unlocked attributes can be edited before the symbol is placed—without the attributes table—in the drawing area.

These *Attribute tables* must be created for all symbols whose attributes should be listed in the **Insert Symbol** dialog box or in the **Symbolmanager**.

All *Symbol attributes* can have predefined values listed in so called value tables that are described in the *Symbols* chapter.

Before you assign *Symbol Attributes* you should read the section *Creating Value Tables* in the *Symbols* chapter

Assigning Symbol Attributes

With the **Symbol Attributes** tool you place a small *attribute table* besides an object that marks the object as a symbol.

If you open a marked symbol with the **Insert Symbol** command or the **Symbolmanager** all attributes contained in this table will be listed in the **Symbol** dialog box. If you open it with **Open** command in the **File** menu the attributes table is brought into the drawing area together with the symbol.

1. Choose the **Symbol Attributes** tool in the ***BOM*** *Tool Palette*.
2. Click to place the **Symbol Attributes** table in an empty spot in the drawing area.

   A table with all assigned attributes is created. If the drawing contains several objects this attributes table is related to all objects in the drawing.
3. Save the object with the **Symbol Attributes** table.

   When you place this symbol in a drawing using the **Insert Symbol** command or the **Symbolmanager** the assigned values can be changed in the **Symbol** dialog box.

Editing Attributes

Assigned attributes are displayed in the *Status Line*, where the default values can be edited.

Show attributes

1. In the **BOM** Tool Palette, select the **Attribute Selection** tool.
2. Move the mouse pointer over the drawing area. As soon as the pointer comes near an object with assigned attributes, these *attributes* and their respective *values* are automatically displayed in the Status Line.

Important*: Attributes must be visible in the* ***Define Attributes*** *dialog box and the proper layout must be active, as indicated in the* ***Define Attributes*** *dialog box, in order to display assigned attributes in the Status Line.*

Changing the values of attributes

Values of attributes can also be edited in the **Edit Objects** dialog. To do so you select the object, choose **Edit objects** in the **File** menu, change the value in the **Edit Objects** dialog and click the **Apply** button.

1. In the **BOM** Tool Palette, select the **Attribute Selection** tool.
2. Move the mouse pointer near an object with assigned attributes.
   The attributes and their respective values are displayed in the Status Line.
3. Select the object with the **Attribute Selection** tool.
4. Change the respective values in the Status Line. To remove an attribute value, delete the value in the entry box.
5. Press the **Enter** key (*Macintosh:* ***Return*** *key*).
   The changed values are assigned to the selected attribute.

Removing attributes

To delete an object with assigned attributes you have to press the **Delete** key twice. Pressing the first time deletes the attributes and pressing the second time deletes the object.

Note: If you want to remove only selected attributes, you have to remove all attributes first and assign than the desired attributes again.

1. In the **BOM** Tool Palette select the **Attribute Selection** tool.
2. Move the mouse pointer near to an object with assigned attributes.
   The attributes and their respective values are displayed in the Status Line.
3. Select the object with the **Attribute Selection** tool.
4. Press the **Ctrl** key and the **Enter** key simultaneously (*Macintosh:* ***Option*** *key and the* ***Return*** *key*).
   All attributes are removed.

Deleting attributes

Values of attributes can also be deleted in the **Edit Objects** dialog.

1. In the **BOM** Tool Palette select the **Attribute Selection** tool.
2. Move the mouse pointer near to an object with assigned attributes.
   The attributes and their respective values are displayed in the Status Line.
3. Select the object with the **Attribute Selection** tool.
4. Select the with the mouse button pressed the attribute *value* you want to delete.
4. Press the **Del** key
4. Press **Enter** key (*Macintosh:* ***Return*** *key*).
   The selected attribute value is deleted.

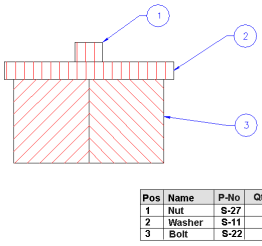
Assigning additional attributes

1. In the **BOM** Tool Palette select the **Attribute Selection** tool.
2. Move the mouse pointer near an object with attributes assigned.
   The attributes and their respective values are displayed in the *Status Line*.
3. Select the object with the **Attribute Selection** tool.
4. Choose the **Attributes** command from the **BOM** submenu in the **Modules** menu.
   The **Define Attributes** dialog box is displayed.
5. Click the **New** button.
6. Define the new attribute by specifying its *name*, *value* and *format*.
7. Click the **Redefine** button.
   The new attribute is displayed in the *Status line*.
8. Change the value of the attribute if necessary.
9. Press **Enter** key (*Macintosh:* ***Return*** *key*).

If you don't want the new attribute to be assigned automatically to each selected object, select the **New** attribute in the list box and click the **Hide** button.

Assigning Item Numbers

The **BOM** tool palette allows you to assign item numbers to objects which will be automatically included in the ***BOM*** *table*.



Pos	Name	P-No	Qty.
1	Nut	S-27	1
2	Washer	S-11	1
3	Bolt	S-22	1

Do not use the **Balloon** tool from the **Dimension** palette to assign *Item Numbers* for objects that you want listed in the *Bill of Materials*. Item Numbers created with the **Balloon** tool are not recognized by the *Bill of Materials* utility.

Item numbers are assigned with the **Item Number** tool in the **BOM** Tool palette.

Item Numbers

With this tool in the **BOM** palette you assign *Item Numbers* to objects. These *Item Numbers* will be automatically included in the *BOM*.

Assigning Item Numbers

1. Select the **Item Number** tool in the **BOM** palette.
   The two entry fields **Item** and **Width** are displayed in the *Status Line*.
2. Enter into the **Item** field the starting item number for the selected object.
3. Click with the **Item Number** tool near the object you want to label.
4. Click with the **Item Number** tool in the drawing area to indicate the position of the *Item Number* balloon.
   The *Item Numbers* are placed at the location you clicked.

Item Numbers can also contain alpha characters as **A**, **B**, **C**, etc.

If you place additional position balloons the *Item Numbers* are automatically increased. The default Item Number is always incremented from the last Item Number used.

If you assign an *Item Number* to an object which was already assigned to another object, the *item numbers* are adjusted as follows:

- If the new *Item Number* is lower than the highest already assigned *Item Number*, all higher *Item Numbers* will be increased.
- If the new *Item Number* is higher than the highest assigned *Item Number*, the existing *Item Numbers* remain unchanged.

The **Item Number** tool automatically recognizes if an object is a copy of an object to which a Item Number is already assigned as long as the original object has an attribute of the type **Quantity**. The **Item Number** tool automatically assigns the identical Item Number of the original object to the object copy.

You cannot assign more than one *Item Number* to an object; if you do, the original *Item Number* is deleted. To get the original Item Number back, do an **Undo** twice.

If you change the Item Number of an object with multiple instances in the **Edit Objects** dialog box, the Item Number of all copies will automatically be adjusted

Note: *Item Number balloons are removed when copying an object.*

Adjusting Item Number balloon and text size

The following procedure describes how to adjust the **Item Number** balloon and **Text** size *before* you assign any Item Numbers.

1. Select the **Item Number** tool in the **BOM** palette.
   The entry fields **Item** and **Width** are displayed in the Status Line.
2. Press the **Tab** key twice and enter the desired width for the *Item Number* balloon. The units for the indicated width are based on the unit specified in the **Units** dialog box from in the **Preferences** submenu of the **Layout** menu.
3. Press **Enter** key (*Macintosh:* ***Return*** *key*).
   The size of the Item Number balloon is now set.
4. Select the **BemSize** command from the **Preferences** submenu in the **Dimension** menu.
5. Select the desired text size.
   The text size of the *Item Number* balloon is now set.

An Item Number's *font* is specified in the **BemText** dialog box of the *Dimension editor*.

Editing Item Numbers

1. Select the desired Item Numbers.
2. Select the Edit **Objects** command from the **Edit** menu.
3. Enter the new *Item Number* in the **Text** field.
4. Click **Apply**.

   The *Item Number* of the selected *Item Number* balloon will be updated accordingly. All higher *Item Numbers* will be adjusted automatically.

Editing the Size of the Item Number Balloon

1. Select the desired Item Number balloon.
2. Select the Edit **Objects** command from the **Edit** menu.
3. Enter the new diameter in the **width** field.
4. Click **Apply**.

   The diameter of the selected *Item Number* balloon will be updated accordingly.

Editing the Text size

An Item Number's *font* is specified in the **BemText** dialog box of the *Dimension editor*.

1. Select the respective *Item Number* balloon.
2. Select the Edit **Objects** command from the **Edit** menu.
3. Enter the new **Text Size** into the in the **Size** field.
4. Click **Apply**.

   The text size of the selected *Item Number* balloon will be updated accordingly.

Item Number text size can be changed dynamically by selecting the Item Number balloon and changing the **BemSize** in the *Dimension Editor*.

Deleting Item Numbers

1. Select the desired Item Number balloon.
2. Select the Delete command in the **Edit** menu or press the **Backspace** key (*Macintosh:* ***Delete*** *key*).

   The selected Item Number will be deleted.

Renumber existing Item Numbers

1. Select the **Item Number** tool in the **BOM** Tool Palette.

   The entry fields **Item** and **Width** are displayed in the Status Line.
2. Enter in the **Item** field the new starting number for the *Item Number*.
3. Press the **Ctrl** key (*Macintosh:* ***Option*** *key*) and keep it pressed.
4. Click the Item Number balloons in the desired order.

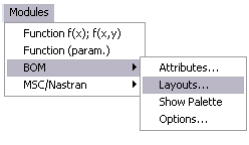
   The Item Number balloon will split. The upper half of the symbol shows the old Item Number and the lower half of the symbol the new Item Number.
5. Release the **Ctrl** key (*Macintosh:* ***Option*** *key*).

   All Item Numbers will be renumbered.

Bill of Materials Layouts

With a ***BOM*** *Layout* you define which attributes will be extracted when creating a ***BOM*** *Table*.

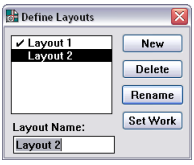
Layouts are defined with the **Layout** command from the **BOM** submenu in the **Modules** menu.



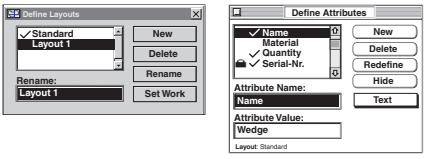
Layouts

This command allows to specify in combination with the **Define Attributes** dialog box different *BOM Layouts*.

The **Layouts** command displays the following dialog box:



Layouts differ only by the attributes they relate to. A layout contains all the visible attributes listed in the **Define Attributes** dialog box. (Visible attributes are indicated by a checkmark.) Therefore, the **Define Attributes** dialog box should be displayed when defining a layout.



Defining in new layout

1. Choose the **Attribute** command from the **BOM** submenu in the **Layout** menu.

   The **Define Layouts** dialog box appears.

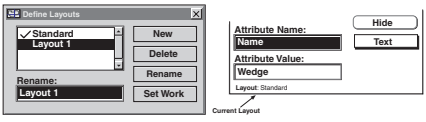
2. Define all the attributes you want to use in different layouts.
3. Make all the attributes visible that you want to save under one layout name.
4. Choose the **Layout** command from the **BOM** submenu in the **Layout** menu.

   The **Define Layouts** dialog box appears.

5. Click the **New** button.
6. Rename the layout in the **Layout Name** entry box.
7. Click the **Rename** button.
8. Click **Set Work**.

   The new layout is now the current layout.

   In the **Define Attributes** dialog box the name of the current layout is displayed in the lower left corner of the dialog box.



9. Click in the title bar of the **Define Attribute** dialog box to activate it..
10. Click the **Save** button.

    All active attributes in the **Define Attribute** dialog box will automatically be saved under the current layout name for the current DraftBoard session.

Editing Layouts

1. Choose the **Layout** command from the **BOM** submenu in the **Layout** menu.

   The **Define Layout** dialog box is displayed.

2. Select the layout you want to edit.
3. Click the **Current** button.
4. Choose the **Attribute** command from the **BOM** submenu in the **Layout** menu.

   The **Define Attribute** dialog box is displayed.

5. **Show** or **Hide** the attributes you want to show or hide (visible attributes are indicated by a checkmark).

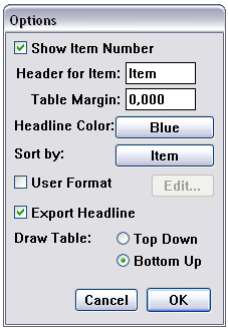
   As soon you make any changes in the **Define Attributes** dialog box a ***** is displayed behind the name of the current layout. When you save the changes made the *****disappears.

If you want to save layouts permanently, you must choose the **Save Preferences** command from the **Preferences** submenu in the **Layout** menu.

Layout Options

For all BOM layouts you define you can set some preferences globally.

When you select the **Options** command from the **BOM** submenu in the **Modules** menu the following dialog box is displayed:



The **Options** dialog box contains the following items:

Show Item Number — Marking this option will create an **Item** column as first column.

Header for Item — In this entry field you name the **Item** column.

Table Margin — In this field you can enter a value for the margin between the text and the vertical column guides.

Headline Color — In this ***pop-up*** *menu* you choose a color for the title text.

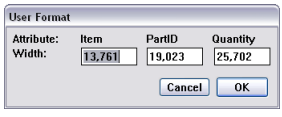
To choose a color you click into the ***pop-up*** *menu* and ***drag*** to the desired color. As soon you release the mouse button the selected color is displayed in the pop-up menu.

Sort by — In this ***pop-up*** *menu* you select the attribute by which the Bill of Materials should be sorted. Click into the pop-up menu with the left mouse button pressed and ***drag*** to the desired attribute. As soon as you release the mouse button, the selected attribute is displayed in the pop-up menu.

User Format — When you mark this option the **Edit** button will be activated.

Edit — Clicking the **Edit** button displays a dialog box where you can define the column width of the *Bill of Materials* table.

All values you specify are based on the current units specified in the **Units** dialog box in **Preferences** submenu of the **Layout** menu.



In this dialog box all attributes of the actual layout are listed. For each attribute the respective column width is displayed. The displayed values for the column widths were calculated by the word length of each attribute and the margin you indicated. You can change the width for each column individually.

Draw Table — These radio buttons let you select whether the Bill of Materials will be drawn from the **Top Down** or from the **Bottom Up**.

The order by which the attributes are displayed in the *Bill of Materials* is determined by the order the attributes were defined in the **Define Attribute** dialog box and cannot be changed in this dialog.

Creating Bill of Materials

After having assigned *attributes* and *item numbers* and having selected a *layout format* you can create the *Bill of Materials*.

Bill of Materials will extract all *item numbers* and their associated *attributes* of the current model and will place a sorted table on the current sheet or export it optionally in a file for later use in other programs.

Pos.	Name	M-Nr.	Qty.	Area
1	Bush	M 237	1	2.34564
2	Screw	M 79	2	
3	Flange	M 881	11	
4	Cotter	S 238	20	2.3876

The generated *Bill of Materials* is an object in ***DraftBoard*** like any other object geometry. That means it can be moved, copied, deleted and edited.

You create Bill of Materials with the **Bill of Materials** tool in the **BOM** palette.

BOM Tool

This tool generates a *Bill of Materials* of the *current model* and places it on the *current sheet*.

Creating a Bill of Materials for the entire drawing

There must be at least one *non-numerical* attribute assigned to an object in order to make a *Bill of Materials*.

1. Choose the **Show Palette** command from the **BOM** submenu in the **Modules** menu.
   The **BOM** palette is displayed.
2. Select the **BOM** tool.
3. Click in the drawing area to indicate a location where you want to display the *Bill of Materials*.
   The *Bill of Materials* of the current layout will be created at the clicked location. The *Bill of Materials* is created in the font currently selected in the **Text** menu.

The *Bill of Materials* can be moved, copied or deleted like any other object in ***DraftBoard***.

Creating a Bill of Materials of selected objects

1. Select all objects you want to display in a Bill of Materials.
2. Choose the **Show Palette** command from the **BOM** submenu in the **Modules** menu.
   The **BOM** palette is displayed.
3. Select the **BOM** tool.
4. With the **Ctrl** key (*Macintosh:* ***Option*** *key*) depressed, click in the drawing area to indicate where you want to place the *Bill of Materials*.
   A *Bill of Materials* containing the *selected objects* will be created and placed at the location you indicated with the last mouse click.

Editing Bill of Materials

1. Select the *Bill of Materials* with the **Selection** tool.
2. Choose **Ungroup** command from the **Arrange** menu.
   The *Bill of Material* is ungrouped into text and line objects which can be edited.

Exporting Bill of Materials

Attributes assigned to objects in a drawing can be exported into an ***ASCII*** *file* to be used in other programs like text editors or spread sheets.

Exporting Bill of Materials

1. Assign all attributes.
2. Select the **Export** command from the **File** menu.
   The **Export** dialog box will be displayed.
3. Mark the **Attribute** option.
4. Select the option **only selected** if you want to export attributes only from selected objects and not from the entire drawing.
5. Click the **OK** button and provide a file name and location.

The Item Numbers and attributes of the entire drawing or of the selected objects only will be exported into an ***ASCII*** *file*.

CHAPTER

15

3D Basics

3D Models

Three-Dimensional Work Space

Model Views

Creating 2D Working Drawings

3D Basics

The process of moving from *2D* to *3D* geometry creation is relatively seamless, since ***DraftBoard*** offers a smooth transition from **2D** to **3D**. All of the tools someone would use to create *2D geometry* in ***DraftBoard*** would also be used to create *3D geometry*. However, there are some tools, features, and concepts that are specific to 3D such as:

- the Setting of the *3D environment,*
- the concept of *Work Planes*,
- the creation and visualization of three-dimensional *Surface Models* that can be exported as *Volume Models*.

This chapter covers the setup of the three-dimensional work space, multiple model views and the derivation of two-dimensional draftings from these 3D Models.

3D Models

Two-dimensional draftings exists mainly of the geometry that makes up the edges of an object plus some additional informations such as *Hatches*, *Text* and *Dimensions*.

In three-dimensional design, objects that define only the edges of a part are called *Wireframe models*. But *Wireframe Models* are only one type of models used in 3D. In principle exist three different model types:

Wireframe Models define only the edges of a part, but not the surfaces covering the frame or the volume contained in a model.

Surface Models define all surfaces that cover the frame of a model. *Surface Models* can be shaded for visualization and their hidden lines can be removed for a more clear representation of the model.

Solid Models define objects including their surfaces and volume. It is possible to add and subtract solid models or create model sections.

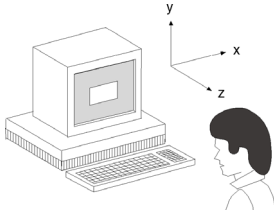
Model Visualizations or *Hidden line Removal* is not possible with *Wireframe Models*.

In ***DraftBoard*** you create *Wireframe Models* with the option to automatically *surface* and *shade* those wireframe models for visualization. These automatically calculated *Surface Models* can be used for *Hidden Line Removal* or exported as *Volume Models* into other programs.

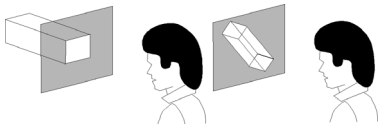
Three-dimensional Work Space

Basically you create three-dimensional objects in ***DraftBoard*** the same way you create two-dimensional draftings. The only difference is that you do not work on a flat area such as a sheet of paper but in an infinitely large three-dimensional work space (with **x**-, **y**- and **z**-axes) in the memory of your computer.

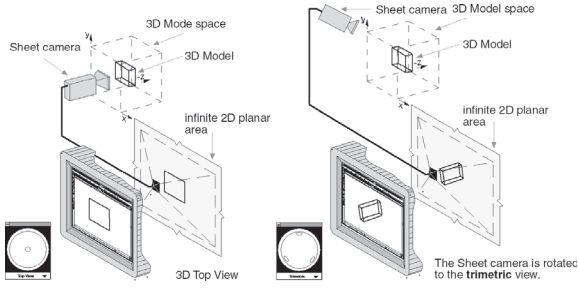
When you open a new drawing in ***DraftBoard*** an empty drawing area is displayed representing a infinite large drawing sheet. The default view for each new drawing is the **Top view**. You are looking down on top of the **x**-, **y**-plane.



To create 3D objects we have to specify in addition to the *width* and the *length* of an object the *height* along the ***z****-direction*, which is coming toward us, away from the screen. To see what we are drawing in the ***z****-direction* we must rotate the image.



As we already know is what we see on the screen not the model itself but only an image of the model, projected by the *Sheet camera* onto the current sheet. The default **Top view** aligns the *Sheet Camera* parallel to the sheet onto the **x**-,**y**-plane.



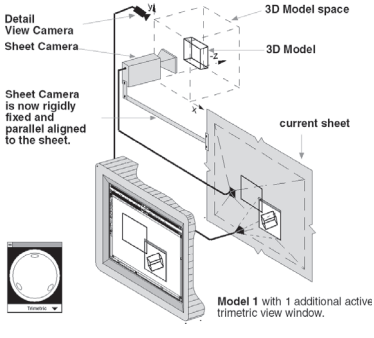
Note*: Rotating a view does* ***not*** *change the orientation of the work plane. (See as well under* ***Work Planes*** *in* ***Chapter 16*** *3D Modelling).*

When we now rotate the view using the on-screen *Trackball* or one of the *View commands* we don't rotate the displayed object geometry (that stays fixed, even if you get the opposite impression on the screen) but the *Sheet Camera* around the model. The rotated image of the model is than projected onto the sheet and therefore called *Sheet view*.

Model Views

As long as no *Detail Views* exist, the *Sheet View* can be rotated unrestricted. As soon as *Detail Views* (picked up by *Detail View Cameras*) are projected into *View windows*, views can be rotated only within these view windows. It is not possible any more to rotate the *Sheet View* that is now rigidly fixed and parallel aligned to the sheet in the **Top View**.

Once you create a detail view the *Sheet View* becomes fixed, because the *Sheet Camera* remains now stationary in the original x,y (world) orientation. You, then will not be able to change the view of the sheet as long as *Detail Views* are placed on the sheet.


Detail View Camera
Sheet Camera
3D Model space
3D Model
Sheet Camera is now rigidly fixed and parallel aligned to the sheet.
current sheet
Model 1 with 1 additional active trimetric view window.

You can rotate views infinitely variable using the on-screen *Trackball* or choose one of the *Standard views* from the **Views** submenu in the **Views** menu.

Rotate views infinitely variable

For rotating views manually ***DraftBoard*** provides a *Trackball*, that can you can display with the **Show Trackball** command in the **Views** menu.

Show/Hide Trackball

This command in the **Views** menu toggles he display of the *Trackball* on the screen.

Rotating the active view

1. Choose the **Show Trackball** command from the **Views** menu.

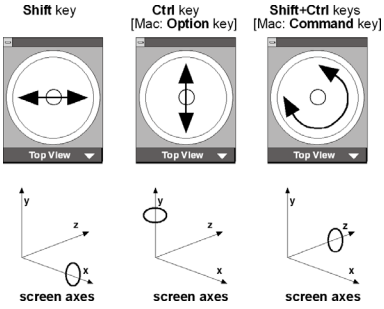
The **Trackball** can be ***dragged*** around the screen.

2. ***Drag*** the pointer on the *Trackball* to rotate the view.
   The model rotates as you ***drag***. The model rotates around the center of the active view or sheet. The movement of the pointer on the trackball corresponds to a fulcrum. The location where you press the mouse button becomes the fulcrum and the movement of the mouse rotates the model around that fulcrum point. What you see on the screen responds to the view of the *Sheet Camera* or *Detail View Camera*.

While rotating a view with the *Trackball* a *Triad* symbol is temporarily displayed at the origin, illustrating the orientation of the ***x***, ***y***, and ***z*** *axis* and the *Work Plane*. See as well under *Triad* in ***Chapter 16***.

Rotating Views

This feature enables you to lock the *Trackball* along two screen axes so you can rotate the view of your geometry around the third screen axis.



The screen axes are different from the axes of your geometry unless you set your view to **Top** with the *Trackball* menu. This makes the screen axes the same as the axes of the block.

Rotating the Screen view with the Trackball vertically

When pressing the **Shift** key and ***dragging*** the cursor on the *Trackball* up or down, the geometry will rotate around the screen ***x****-axis* (vertically).

Rotating the Screen view with the Trackball horizontally

When pressing the **Ctrl** key *(Macintosh:* ***Option*** *key)* and ***dragging*** the cursor on the *Trackball* left or right, the geometry will rotate around the screen ***y****-axis* (horizontally).

Rotating the Screen view with the Trackball circular

When pressing the **Shift+Ctrl** keys *(Macintosh:* **Command** *key)* and ***dragging*** the cursor on the *Trackball* in a circular motion, the geometry will rotate around the screen ***z****-axis* (circular).

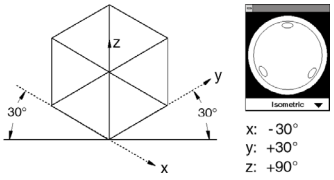
Selecting Standard Views

DraftBoard provides the following predefined, orthogonal standard views that offer a perfect environment for creating 3D wireframe models:

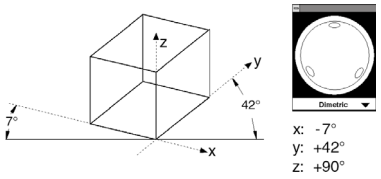
Top	View of the x,y plane from above.
Bottom	View of the x,y plane from below.
Front	View of the x,z plane from the front.
Back	View of the x,z plane from the back.
Right	View of the y,z plane from the right.

Left View of the y,z plane from the left.

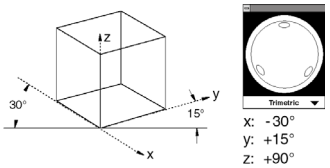
Isometric The view of the axes is rotated as shown:



Dimetric The view of the axes is rotated as shown:



Trimetric The view of the axes is rotated as shown:



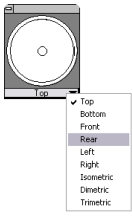
Trackball View menu

The *Trackball* has a pull-down menu for specifying a *Standard View* of the current view window or the sheet, if there are no view windows.

Selecting a Standard View with the Trackball menu

1. Move the pointer to the current view name displayed at the bottom of the *Trackball* menu.
2. Press the mouse button.
   The *Trackball menu* is displayed.

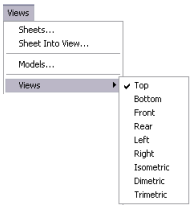
With the *Trackball* you can rotate the view in an active view window or the *Sheet view* as long as there are no detail views.



3. ***Drag*** the pointer to the view orientation you want to display and release the mouse button.
   The current view orientation is updated.

Views

This command in the **Views** menu displays the view orientation submenu so that you can set the orientation of the active view or the sheet as long as there are no detail views.



Defining Views

There are several possibilities to define views:

- using the **Define View** command in the **Views** menu.
- using the **Unfold View** command in the **Views** menu.
- using the **Flatten View** command in the **Views** menu.
- using the **View the Plane** command in the **Views** menu.

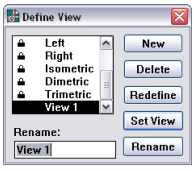
Define View

This command in the **Views** menu allows you to define auxiliary view orientations or change the standard view orientations such as the *Top View*.

The *Standard Views* cannot be changed unless the locked checkbox is clicked off.

All views defined additionally using the **Define View** command are automatically included into the **Views** submenu and can be selected from the **Trackball** menu.

Choosing the **Define View** command displays the following dialog box:



The **Define View** dialog box allows the following settings:

View list This list window contains all defined views. A check-mark indicates the active view, a locked checkbox any locked view. To redefine a view the locked checkbox has to be clicked off.

New The **New** button displays the **Redefine View** dialog box where you can specify a new view. When you click the **OK** button the dialog box is closed and the new view is added to the views list. You can rename the new view using the **Rename** button.

How to define a view is described in the next **New/ Redefine** section of this chapter.

Delete This button deletes the selected view in the views list. Locked views and the active view cannot be deleted.

Redefine The **Redefine** button displays the **Redefine View** dialog box where you can redefine the selected view. Locked views and the active view cannot be redefined.

How to redefine a view is described in the next **New/ Redefine** section of this chapter.

Set View This button makes the selected view the active view. The active view is indicated by a checkmark.

Rename Using this button you can rename any selected view. Locked views and the active view cannot be deleted.

Activating a View Orientation

1. Select the view you want to activate in the **Define View** dialog box.
2. Click the **Set View** button.

You can select as well a view in the **Views** submenu of the **View** menu.

Renaming a View Orientation

1. Select the view you want to rename in the **Define View** dialog box.
2. Enter the new name in the **Rename** field.
3. Click the **Rename** button.

Locked views and the active view cannot be re-named.

Locking/Unlocking a View Orientation

1. Select the view you want to *lock* or *unlock* in the **Define View** dialog box.
2. Click on the *Lock symbol* to unlock a view and click once in front of a view name in the views list to lock the view. If the view was locked the *Lock symbol* will be removed, if it was unlocked the lock symbol will be displayed.

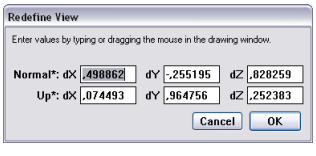
Deleting a View Orientation

Select the view orientation to be deleted from the **Views** list and click the **Delete** button.

Locked views and the active view cannot be deleted.

New/Redefine

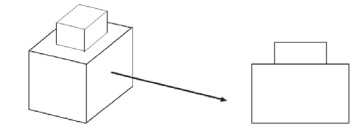
When you click the **New** or the **Redefine** button the **Redefine View** dialog box is displayed:


Redefine View
Enter values by typing or dragging the mouse in the drawing window.
Normal*: dX ,498862 dY -,255195 dZ ,828259
Up*: dX ,074493 dY ,964756 dZ ,252383
Cancel
OK

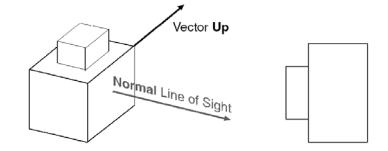
The **Redefine View** dialog box displays always the location of the current view. The current view can be any selected view from the **Views list** or any view specified using the on-screen *Trackball*.

The **Redefine View** dialog box allows the following settings:

Normal* In these entry fields you specify the Normal *(90°) line of sight* by ***dragging*** a vector on the drawing area. The direction you indicate with the ***dragged*** line for the ***Normal*** *line of sight* corresponds with a vector coming toward you, away from the screen, when the view is active.



Up* In these entry fields you specify by ***dragging*** a vector on the drawing area which side of an object in the future view will show **up**wards. The ***dragged*** vector for the **Up** direction of the *Line of Sight* is always rectangular to the ***Normal*** *line of sight* .



OK When you click the **OK** button the **Redefine View** dialog box is closed. If the dialog box was opened using the **New** button, a new view with the name **View 1** will be added to the **Views list**. If the dialog box was opened using the **Redefine** button, the selected is redefined.

Defining or Redefining a View

You can use one of two methods to specify the orientation of the axes of a view:

By example

1. Manipulate the view orientation manually with the *Trackball*.
2. Select the **Define View** command in the **Views** menu.
   The **Define View** dialog box is displayed.
3. Select a view and click the **Redefine** button, if you want to redefine the view with the current values, or click the **New** button, if you want to create a new view based on the current values.
   The **Redefine View** dialog box is displayed showing the values of the current view on the screen.
4. Click the **OK** button.
   The **Redefine View** dialog box is closed and the selected view is redefined or a new view is created with the name **View 1**, that can be renamed using the **Rename** button.

By definition

1. Select the **Define View** command in the **Views** menu.
   The **Define View** dialog box is displayed.
2. Select a view and click the **Redefine** button, if you want to redefine the view with the current values, or click the **New** button, if you want to create a new view based on the current values.
   The **Redefine View** dialog box is displayed. The ***Normal*** *Line of Sight* specification is highlighted.
3. ***Drag*** a vector with the mouse to indicate the ***Normal*** *Line of Sight*.
   The **Up** entry boxes become selected.
4. ***Drag*** a vector with the mouse to indicate the ***Up*** *direction of the Sight*.
5. Click the **OK** button.

15 When you ***drag*** the vector for the Normal Line of Sight, know that the vector from the beginning point, points directly **at** you, not **away** from you. So when this new view is activated, the normal vector points **out** of the screen, not **into** it.

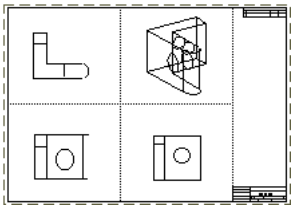
The **Redefine View** dialog box is closed and the selected view is redefined or a new view is created with the name **View 1**, that can be renamed using the **Rename** button.

Designing in multiple Views

As soon as you place any detail view, you can only change the view orientation within a view window, but not the view orientation of the sheet, that is then rigidly fixed in the *Top view*. Therefore it makes sense to design three-dimensional models using multiple view windows, that allow you to view the geometry from different view angles simultaneously.

View Windows are described in detail in *Chapter **10** Structuring Drawings*.

A *View Window* is nothing else than the projection of a *detail view* on the current sheet picked up by a *View Camera* at a particular angle. If the option **Draw View Boundaries** in the **Views** menu is active, the boundaries of all inactive views is shown by a dotted line. The active view displays always its title bar.

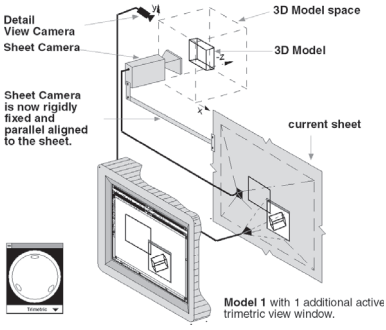


The geometry within view windows is associative, since they are displaying all the same model. When you make a change in one window, the change effects the model and therefore the geometry in all views. There exist two different methods for creating 3D wireframe models:

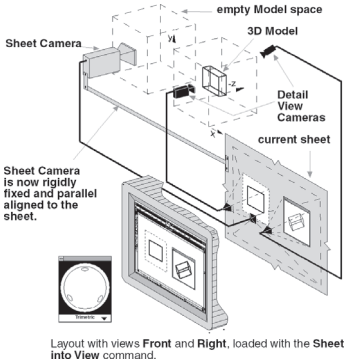
1. Start the model creation on the drawing sheet and add views later using the **Detail View** tool for editing and viewing the model from different angles.
2. Use a so called *View Layout* from the beginning and create the model in different view windows simultaneously.

Using the first method you open a new drawing and create the geometry on an empty sheet. With the *Trackball* or one of the *View* commands you can rotate your model and edit it from different angles as long as you have not created any detail view windows.

As soon as you place a detail view windows on the sheet, the *Sheet View* is fixed in the **Top** view, while the detail views can display different view orientations of your geometry.



This method has the disadvantage that the *Sheet View* of your geometry may overlap the view windows on the sheet. To avoid this problem you could assign the sheet to a new model by creating a new **Model 2** using the **Model** command from the **Views** menu, clicking then on the sheet to activate it and making finally **Model 2** the current model.



Layout with views **Front** and **Right**, loaded with the **Sheet into View** command.

You could create as well a new **Sheet** and copy all detail views onto the new sheet to get a clear view on the view windows. To copy a detail view from **Sheet 1** to **Sheet 2**, to you must cut the detail views using the **Cut** command from the **View Window** menu, activate then **Sheet 2** in the **Sheet** dialog box and paste the detail views onto **Sheet 2** using the **Paste** command from the **Edit** menu. *Models*, *Sheets* and *Views* are described in detail in *Chapter **10** Structuring Drawings*.

Something similar, but more elegant is to load a *View Layout* using the **Sheet into View** command in the **Views** menu, since the **Sheet into View** command creates automatically a new model, projects it on the current sheet and puts all geometry into one or more view windows.

View Layouts

The **Layout** folder contains besides *View Layouts* as well *Standard Drawing formats* that you can import or load using the **Sheet into View** command.

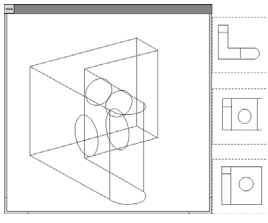
View Layouts allow you to create 3D models simultaneously in several view windows without having the geometry displayed on the current sheet at the same time.

There are two *View Layouts* optimized for 3D Design: **Design 4** and **Draft 4**. Both layouts are stored in the **Layout** folder in the **DraftBoard** folder and can be loaded with **Sheet into View** command.

Design 4 [Design4.vlm]

This layout contains four views. The **Trimetric View** is scaled as specified in the **Sheet into View** dialog box and the other three views are scaled to 25% of the **Trimetric** view.

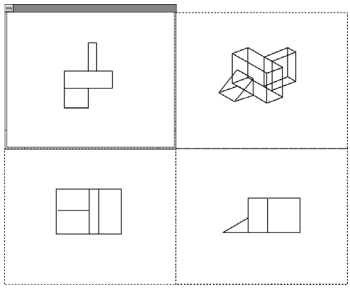
The geometry within view windows is associative, since they are displaying all the same model. When you make a change in one view window, the change effects the model and therefore the geometry in all views. **Important**: *Hatches*, *Text* and *Dimensions* are view dependant and appear only in the views they were created in.



Draft 4 [Draft4.vlm]

This layout creates four full-scale views scaled as specified in the **Sheet into View** dialog box: *Top*, *Front*, *Right* and *Trimetric*.

The **Flatten View** command in the **Views** menu, that is described in a later section of this chapter, allows you to cancel the associativity existing between the displayed view windows.



Creating View Layouts

You might look at one of the files in the **Layouts** folder to see how it is set up. Be sure not to make any unwanted changes to these layout files.

All view layouts available from the Sheet into View dialog box are individual ***DraftBoard*** drawing files stored in the **Layouts** folder in the **DraftBoard** folder.

If you want to create your own layout, simply edit one of the existing drawings in the **Layouts** folder, or create a new drawing and save it in the **Layouts** folder. The file name then will appear automatically in the pull-down menu in the **Sheet into View** dialog box.

Creating custom View Layouts

The easiest way to create customized layouts is to edit existing ones.

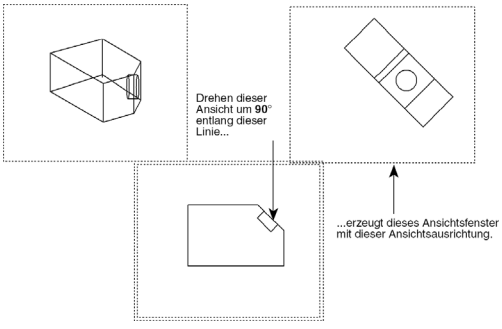
1. Open a layout file in the **Layouts** folder using the **Open** command
   in the **File** menu.
2. Save the file under a different name before you do any changes.
3. Customize the layout to your needs by modifying the *Number*, *Size*, *Scale* or *View Orientation* of the view windows.
4. Save the modified layout in the **Layouts** folder.
5. Select the **Sheet into View** in the **Views** menu.
6. The new layout appears automatically in the pull-down menu in the
   Sheet Into View dialog box.

Another way to create your own layout is to design it in a new document.

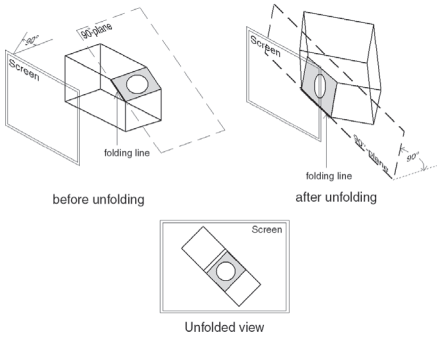
1. Open a new document using the **New** command in the **File** menu.
2. Create a view window with the **Detail View** function.
3. Make the current sheet active by clicking anywhere outside the view window on the sheet.
4. Create a new model using the **Model** command in the **Views** menu.
5. Make the new **Model 2** the current model in the **Models** dialog box.
6. Copy the view window as often as you want with the **Copy** command in the **View Window** menu and the **Paste** command in the **Edit** menu.
7. Set each view window to the desired view orientation with the **Views** commands in the **Views** menu or in the **Trackball** menu.
8. Select the **Properties** command in the detail view window's control menu to scale each view window.
9. Save the new layout under a proper name in the **Layouts** folder.

Unfold View

The **Unfold View** command in the **Views** menu allows you to create a view orientation from the active view window by specifying a line about which to unfold the new view orientation.



By specifying a line you define a **90°**-plane along that line, which is coming toward you away from the screen. This plane is folded by **90°** and creates the new view orientation from the active view.



*Note: You should use the **Unfold View** command only in a view orientation which is aligned parallel to an object face. If you use it in a differently aligned view (object face is not parallel to the screen) this command will work correctly, but the result will be unpredictable in most cases.*

Unfolding a View

1. Select the **Unfold View** command in the **Views** menu.
2. Click the endpoints of the line to unfold on.

   The active view window shows the view unfolded 90° from the line you specify.

You don't need an actual line, you can as well ***drag*** a vector.

View the Plane

This command in the **Views** menu rotates the view orientation in the active view window to match the work plane. The view changes so the work plane becomes horizontal on the screen. The **y**-direction of the work plane becomes *vertical*, and the **z**-direction of the work plane becomes the *line of sight*, coming directly out of the screen.

To leave this view orientation, select a view from the **Trackball** menu or the **Views** submenu, or use the *Trackball* for free rotation.

Editing Views without editing the Model

The geometry within view windows is associative. When you make a change in one window, the change affects the model and therefore the geometry in all views. In the drafting phase of the design process, you may want to edit the geometry in one view window without making the same changes to the model.

You can accomplish this by using the called **Flatten View** command on the **Views** menu. When you flatten a view, the geometry is taken out of the view and placed back on the sheet, where it is no longer associated with the model. In this way, you can make changes to the geometry without changing the model or other views.

Flatten View

Important: If you flatten a view containing a *surfaced model* displayed in the *hidden line mode*, not only the visible object edges will be flattened but the *original* geometry as well.
The visible edges will be placed on the current layer while the original geometry will be placed on the layer **Wireframe**. The layer **Wireframe** will be automatically created and hidden. It must be displayed for the dimensioning of circles, since it is not possible to dimension the visible edges but only the original geometry. See as well under *Dimensions in 3D* in **Chapter 16** *3D Modelling*.

This command places a projection of the visible geometry in the active view window onto the sheet at full scale. When the view is flattened, all overlapping lines of equal length and all lines parallel to the line of sight are removed. The geometry is no longer associated with the model. If you make changes to the flattened geometry, those changes do not affect the model. If you make changes to the model, the flattened geometry is not changed.

When you flatten a detail view, you are flattening only what is visible within the detail view window. Objects partly visible in a view window will be clipped, objects outside of a view window will be ignored.

Text and *Crosshatching* are treated like geometry by this command. *Dimensions* will no longer update if you make a change to the 3D model since the flattened geometry is disassociated, but they will change if you alter the flattened geometry.

Also if you dimension your geometry after it is flattened, you will have to enter the desired values because DraftBoard will simply read the length of the projected flat lines. If you want to keep the dimensions, copy the view and flatten the copy, or group the geometry with the dimensions, then flatten the view.

If the following message displays, ***The current view and the draft view have the same model***, you must change the model on the sheet. Click the sheet, outside all views, then create a new model using the **Models** command in the **Views** menu, then click **Current** to activate the new model, and then flatten the view again.

Flatten a View

1. Select a view window.
2. Choose the **Flatten View** command in the **Views** menu.

   A dialog box displays a warning message. Since flattening disassociates the current view from the model, you are given this opportunity to be certain you have selected the view you want to flatten and that you really want to flatten the selected view.

   All dimensions in the view are deleted because the 3D dimensions would not be correct in 2D geometry.
3. Click the **OK** button.

   The geometry is placed on the sheet at full scale.

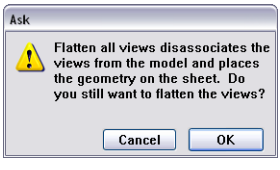
Flatten all views

Often **Detail views** and the **Sheet view** show different models. When exporting drawings only one model can be exported, therefore detail views can get lost.

When exporting ***DXF/DWG*** *Files* not all views have to be flattened, since several views are automatically detected during export and a dialog box with corresponding options will be displayed. See as well *Exporting* ***DXF/DWG****-Files* in this manual.

This can be avoided with the **Flatten all Views** command.

When you select this command the following dialog window is displayed:


Ask
Flatten all views disassociates the views from the model and places the geometry on the sheet. Do you still want to flatten the views?
Cancel
OK

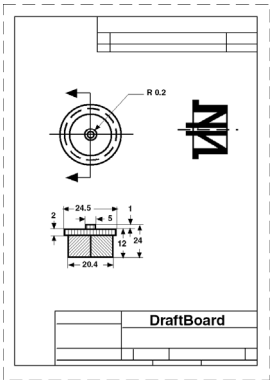
This note warns you that all detail views will be disassociated from the model and therefore will loose their associativity.

When exporting a drawings with more than one model, using the export formats **EMF**, **WMF**, **PNG**, **DXF/DWG** you will automatically be offered to flatten all views if you want to export all detail views.

Creating 2D Working Drawings

Even though ***DraftBoard*** allows to create 3D Models intuitively, it has not eliminated the need for drafting. While modifying, storing, and sharing models are the basis of computer design, drafting is still essential. ***DraftBoard*** makes drafting painless. You can produce finished drawings for proposals, planning, and patents quickly and efficiently.

You can create one model, choose a single command, and voilá— a four-view layout, scaled perfectly!



You can create two-dimensional draftings of a 3D model by importing a *Standard Drawing Format* or by loading a *Standard Drawing Format* with one or more views using the **Sheet into View** command.

Placing a 3D Object in a Standard Drawing Format

1. Create the 3D Object on the *Sheet* or using a *View Layout* with one or more views.
2. Import a *Standard Drawing Format* from the **Layouts** folder in the **DraftBoard** folder using the **Import** command in the **File** menu. Mark the option **unscaled** in the **Import Option** dialog box so the drawing format is imported at its original size.
3. Align the geometry within the view windows and the view windows within the drawing format according to the standard drawing rules.
4. Flatten a view in case you want to edit the geometry in that view window without modifying the geometry in the other view windows.
5. Dimension the geometry on the sheet or in the view windows.

As soon as you place a detail view windows on the sheet, the *Sheet View* is fixed in the **Top** view, while the detail views can display different view orientations of your geometry.
How to import a drawing format is described in ***Chapter 9*** *Basic Drafting.*

Since each view is associated with the model all changes made in one view will be reflected in the other views. The same is not true for *Text*, *Dimensions*, *Crosshatching* and *Fills*.

Creating a 2D Drafting using the Sheet into View command

1. Create the 3D Model on the Sheet .
2. Load a *Standard Drawing Format* with one or more views using the **Sheet into View** command in the **Views** menu.
3. Align the model within the view windows and the view windows within the drawing format according to the standard drawing rules.
4. Flatten a view in case you want to edit the geometry in that view window without modifying the geometry in the other view windows.
5. Dimension the geometry on the sheet or in the view windows.

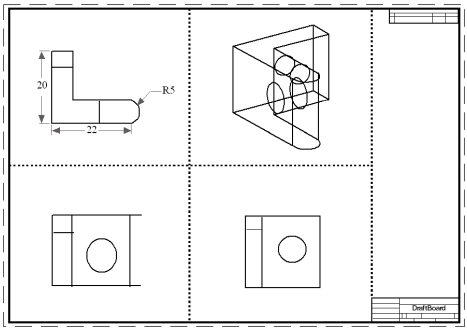
It is recommended to delete all existing detail views before using the **Sheet into View** command.

Since each view is associated with the model all changes made in one view will be reflected in the other views. The same is not true for *Text*, *Dimensions*, *Crosshatching* and *Fills*.

The following Standard Drawing Formats are available in the Layouts folder for the use with 2D Draftings:

A4 Portrait & 4 Views	**A1 Portrait & 4 Views**
A4 Landscape & 4 Views	**A1 Landscape & 4 Views**
A3-Portrait & 4 Views	**A0 Portrait & 4 Views**
A3 Landscape & 4 Views	**A0 Landscape & 4 Views**
A2 Portrait & 4 Views	**A2 Landscape & 4 Views**

The object geometry on the sheet will be placed in four views (*Top*, *Front*, *Right* and *Trimetric*) within a Standard Drawing Format (A4, A3, A2, A1, A0).



CHAPTER

16

3D Modelling

Work Plane

The Design Process

3D Features and Tools

3D Construction Rules

3D Modelling

This chapter provides all information you need for modelling three-dimensional objects:

- Work Planes
- Design Process
- 3D Construction Tools
- Rules for 3D Construction

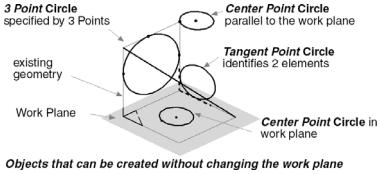
Work Plane

In traditional 3D computer-aided design, you work as though you have propped a piece of glass on one of the faces of the model and draw on that. Every time you want to work on a different plane, you have to pick up your glass and carry it to the new position. With ***DraftBoard***, you can *move* around inside the model and draw wherever you want.

Generally, all you have to worry about is the design, not moving the piece of glass. Due to the *Drawing Assistant* the work plane in ***DraftBoard*** is rarely used. But when it's required you only have to specify the *orientation* of the work plane but not its *position* like in conventional CAD-programs.

The *Drawing Assistant* keeps tabs on the angles and intersections for you in all three dimensions. It identifies the plane of an object automatically as soon as it snaps on one of the objects construction points. Therefore you can, if geometry exists, draw relative to the identified plane.

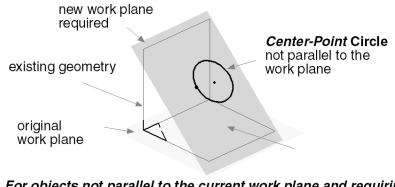
Also for objects you create with tools requiring *three construction points* such as the ***3-Point Circle*** tool or ***Tangent-Point Circle*** tool, the orientation of the work plane must not be changed.



Objects that can be created without changing the work plane

When a tool creates geometry from two points, the geometry will lie on a plane parallel to the work plane. If the tool uses three points to define the geometry, the geometry will be on the plane defined by the three points.

Only if you are creating geometry that is not parallel to the current work plane, and if you are using a tool that requires only *two points* for creation, such as the ***Center-Point Circle***, ***Rectangle***, or ***2-Point Ellipse***, you need to modify the orientation of the work plane to properly place the geometry. The exception to this rule are ***Lines*** and ***NURB Splines***, that can be created independent from the orientation and location of the current work plane.

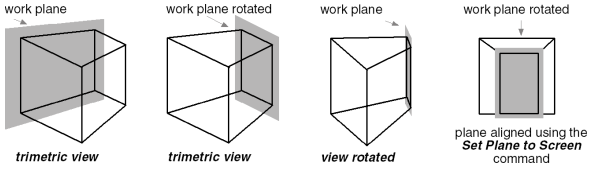


For objects not parallel to the current work plane and requiring two points for creation the orientation of the work plane must be modified.

Differences between Work Plane and Views

When you change the orientation of the *Work Plane*, the *View* orientation remains unchanged. The view orientation specifies the viewing angle, the direction you look at your geometry, while the orientation of the work plane determines the plane on which you create your geometry.

But rotating a view can bring the work plane into a position that makes it impossible to create any geometry on it.



The command **Set Plane to Screen**, that keeps the *Work Plane* always *parallel* to the *Screen* while rotating a view and the command **View the Plane** that aligns the *View* *parallel* to the *Work Plane* help to avoid this problem.

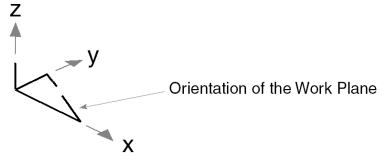
Displaying the Work Plane

When you open a new drawing in ***DraftBoard***, the default orientation of the work plane is **Top**. You can display a *Triad* symbol on the screen to control the orientation and position of the work plane.

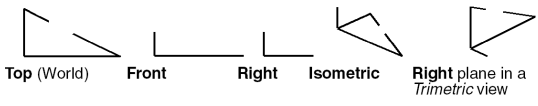
Show/Hide Triad

This command displays the *Triad* symbol in the upper-left corner of the screen or active view window to illustrate the orientation of the ***x***, ***y***, ***z*** *axis* and the *work plane*.

If you want to see the *Triad* at the **Origin** (**0**,**0**,**0**), set the **Grid Spacing** to **0** in the **Preferences** submenu in the **Layout** menu.



Possible representation of the Work Plane depending on the selected View

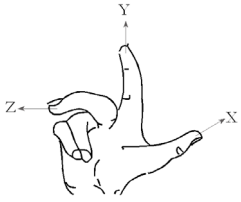


The *Work Plane* is represented by an *open triangle* within the three axes **x**, **y** and **z**. The displayed orientation of these axes is depending on the selected view orientation.

If you have not created a *detail view*, the triad symbol displays in the upper-left corner of the sheet. Once a view window is created, it displays in the upper-left corner of the active view window, since the sheet view cannot be rotated any more; it remains stationary in the *World Coordinate System* at the *Top* orientation.

The *Triad* symbol also temporarily displays at the origin when you rotate the view orientation manually with the *trackball*.

The *Triad* displays the principle of the ***Right-Hand Rule***—a memory aid for the relative directions of the positive axes. With your right palm upturned, the **Thumb** (**X**) points right, the **Index Finger** (**Y**) points straight ahead, and the **Middle Finger** (**Z**) points up. If you move your hand to indicate the **x**- and **y**-axes, you can easily see the direction of the ***z****-axis*.



The Right Hand Rule

Selecting Standard Planes

DraftBoard offers several commands to define the orientation of the work plane and three predefined *Standard Planes* to choose from. The three *Standard Planes* are predefined as follows:

Front **x**- ,**z** Plane

Right **y**- ,**z** Plane

World (Top) **x**- ,**y** Plane
Orientation and *Origin* of **World** plane was specified together with the *Model Space*. The **World** (*Top*) plane is the default plane for any new drawing and can **not** be modified.

If you have used other computer applications, you may be familiar with the terms *World Coordinate System* and *User Coordinate System*. The **World** *(Top)* work plane of ***DraftBoard*** is equivalent to the *World Coordinate System* and any ***nonstandard***, *temporary* work plane is equivalent to the *User Coordinate System*.

Planes

This command in the **3D** menu displays the **Planes** submenu so that you can set the current work plane. Standard work planes (*Top*, *Right* and *Front*) are always displayed, as well as any *temporary* plane you have generated and any planes you have created with the **Define Plane** command in the **3D** menu.

Setting the Work Plane

1. Select the **Planes** command in the **3D** menu.
   The **Planes** submenu is displayed:

2. Select one of the displayed planes.
   The work plane in the sheet or active view window is rotated as specified, and the work plane name is checked in the submenu.

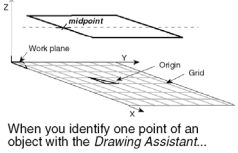
Often it is useful to align the view with the current work plane using the **View the Plane** command in the **Views** menu.

If you create a work plane with the **3-Point Plane** command it is named Temp and will exist only for the duration of the current ***DraftBoard*** session. You can rename the **Temp** work plane with the **Define Plane** command to make it permanent. There can be only one plane called **Temp Plane** in a document. If another plane is created using the **3-Point Plane** command without changing the name of the first, the new **Temp Plane** replaces the old one.

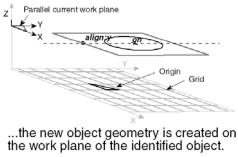
Specifying the position of the work plane

In conventional CAD-programs you have to specify both the *Orientation* of the work plane **and** its exact *Location* along the **z**-axis.

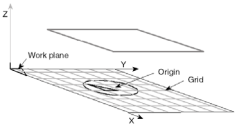
In ***DraftBoard*** you need only to specify the *Orientation* of the work plane. Once that is done, then all planes parallel to that orientation act equally as the current work plane, e.g. the *Drawing Assistant* identifies the *Location* of the work plane automatically.



When you identify one point of an object with the *Drawing Assistant*...



...the new object geometry is created on the work plane of the identified object.



If you do not identify an object the new object geometry is placed onto the work plane at the origin.

To display the **Grid** with **Origin** select **Show Grid** in the **Layout** menu.

Another way to explain the relationship presented in the graphics above, is to remember the role of the *Drawing Assistant*. For new geometry to be placed on the same plane, you need to brush over that "old" geometry to wake up one of its control points like midpoint or endpoint. ***DraftBoard*** will then use retain this plane information as you create the new geometry.

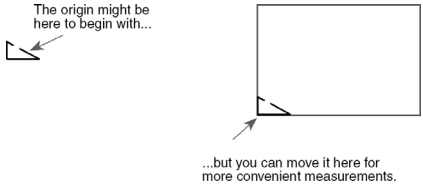
If no object exists or no geometry is referenced for plane information, ***DraftBoard*** places the new object geometry onto the work plane at the origin.

Moving the origin

Relative Coordinates also called *User Coordinates* are very useful if you want to draw relative to existing objects. The **Set Origin** command allows you to move the *Coordinate Reference Point* to another location.

The current position of the origin will be only visible when the *Grid* is displayed using the **Show Grid** command in the **Layout** menu.

When you open a new drawing the *Origin of the Drawing* is identical with the absolute origin of the *World Coordinate System* (**0**,**0**,**0**). To display relative coordinates in the *Position Indicator* you can move the origin of the work plane using the **Set Origin** command. With the **3-Point Plane** command you move the origin of the current work plane and change its orientation.



As soon as you select the **World** (Top) plane again from the **Planes** submenu of the **3D** menu you return to the *World Coordinate System* and move the origin of the work plane to absolute zero (**0**,**0**,**0**).

Set Origin

This command sets a new origin for the current work plane.

Specifying a new origin

1. Choose the **Set Origin** command from the **3D** menu.
2. Click in the drawing area to indicate the location for the new origin.

   The origin of the current work plane is moved while the orientation of ***x***, ***y*** and ***z*** *axes* remains the same.

Setting a new Work Plane

When using certain tools you have to define a work plane other than the standard planes. For example, if you want to create an object on an angled face, using a tool that defines an object by only two points.

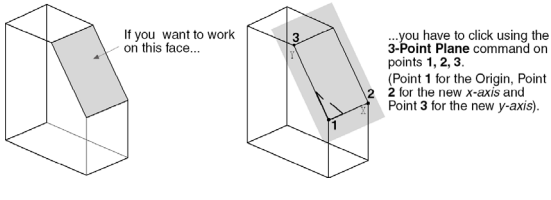
You can define a new plane with two commands on the **3D** menu: ***3*-Point Plane** and **Define Plane**. With the **3-Point Plane** command you set a new work plane, that you can name and save using the **Define Plane** command.

3-Point Plane

You just must follow the directions on the *Message Line* when using the **3-Point Plane** command.

Be aware that modifying the orientation of the work plane doesn't automatically mean that you will work on a visible face. The view orientation must be adjusted accordingly.

This command in the **3D** menu sets a ***temporary*** work plane by simply indicating the *Origin* and the *horizontal* (***X***) and *vertical* (***Y***) directions.


If you want to work on this face...
3
2
1
...you have to click using the 3-Point Plane command on points 1, 2, 3.
(Point 1 for the Origin, Point 2 for the new x-axis and Point 3 for the new y-axis).

Specifying a temporary Work Plane

1. Choose the **3-Point Plane** command from the **3D** menu.
2. Click in the drawing area to indicate the point you want for the **Origin** of the new work plane.
3. Click a point to define the positive **x-axis**.
4. Click a point to define the positive **y-axis**.

A temporary work plane is created and named **TempPlane**. The work plane is moved, but the view does not change.

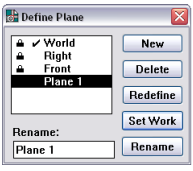
If you do not name the new plane with **Define Plane**, it will be replaced by the next *3-point plane* you specify and will be removed from the list of available planes when you quit ***DraftBoard***.

Define Plane

Allows you to define a new work plane, name a *temporary* plane, or change a standard plane. Standard planes cannot be changed unless you click off the lock icon.

If you change any of the standard planes, you can always revert to the ***World*** *Plane* (**Top** Plane) to return to the original orientation.

When you select the **Define Plane** command from the **3D** menu the following dialog box is displayed:



The **Define Plane** dialog box contains the following items:

Planes List This list box contains all defined planes. The work plane is indicated by a *Checkmark*, all locked planes by a *Lock* icon. Locked planes must be unlocked for editing by clicking the lock icon.

New Clicking this button displays the **Redefine Plane** dialog box, that allows you to define a plane. As soon as you click the **OK** button the dialog box is closed and the new plane **Plane 1** is added to the *Planes List*. Using the **Rename** button you can give the plane a more appropriate name.

How to define a plane is described in the section **New/Redefine**.

Delete This button allows you to delete a selected plane. Locked planes and the current work plane can not be deleted.

Redefine Clicking this button displays the **Redefine Plane** dialog box, that allows you to redefine the selected plane.

How to redefine a plane is described in the section **New/Redefine**.

Set Work This button allows you to make a selected plane the *Work Plane*. The work plane is indicated by a *Checkmark*.

Rename This button allows you to rename a selected plane in the **Rename** box. Locked planes and the current work plane can not be renamed.

Selecting the Work Plane

1. Select the plane you want to make the work plane in the **Define Plane** dialog box.
2. Click the **Set Work** button.

You can select the *Work Plane* also from the **Planes** submenu in the **3D** menu.

Renaming a plane

1. Select in the plane you want to rename.
2. Overwrite the name with anew name in the **Rename** box.
3. Click the **Rename** button.

You cannot rename the **World** (*Top*) plane, all *locked* planes and the current *Work Plane*.

Locking/Unlocking a plane

1. Select the plane you want to *lock*/*unlock* in the **Define Plane** dialog box.
2. Click the *Lock* icon to *unlock* the plane or in front of the name to *lock* the plane.

   If the name was locked the *Lock* icon disappears, if it was unlocked the lock icon is displayed in front of the name.

Deleting a plane

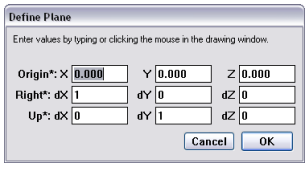
Select the plane you want to delete in the **Define Plane** dialog box and click the **Delete** button.

You cannot delete the **World** (*Top*) plane, all *locked* planes and the *Work Plane*.

New/Redefine

If you click the **New** or **Refine** button in the **Define Plane** dialog box the **Redefine Plane** dialog box is displayed:

You cannot edit the **World** (*Top*) plane, all *locked* planes and the *Work Plane*.



The **Redefine** dialog box displays always the definition values of the current work plane, either of a selected *Standard Plane* or of a *Temporary Plane*.

The **Redefine** dialog box contains the following elements:

Origin* These entry boxes display the coordinates for the *Origin* of the work plane. You can type in new values or indicate the new location of the origin with a mouse click on the drawing area.

Right* The **Right** entry boxes display values for the direction of the ***x*** *axis*. You can type in new values or indicate the new direction of the ***x*** *axis* with the mouse on the drawing area.

Up* The **Up** entry boxes display values for the direction of the ***y*** *axis*. You can type in new values or indicate the new direction of the ***y*** *axis* with the mouse on the drawing area.

OK Clicking the **OK** button closes the **Redefine** dialog box and — in case the **Redefine** dialog box was displayed using the **New** button — a new plane is added to the **Planes** list in the **Define** dialog box.

In case the **Redefine** dialog box was displayed using the **Redefine** button the orientation of the selected plane is redefined.

Specifying a new work plane

Two different methods exist for defining a plane:

By a temporary plane

1. Define a *temporary plane* using the **3-Point Plane** command.
2. Select the **Define Plane** command in the **3D** menu.
   The **Define Plane** dialog box is displayed listing the *temporary plane* in the **Planes** list box.
3. Select the **TempPlane** in the **Planes** list box and rename the plane in the **Rename** entry box.
4. Click the **Rename** button.
5. Click the **OK** button.
   The **Define Plane** dialog box is closed and the new plane is added to the **Planes** submenu in the **3D** menu.

By definition

1. Select the **Define Plane** command in the **3D** menu.
   The **Define Plane** dialog box is displayed.
2. Click the **New** button.
   The **Redefine Plane** dialog box displays and the **Origin** specification is already highlighted.
3. On the drawing area, click the location for the new origin.
   The first **Right** entry box is selected.
4. Click a location on the positive **x-axis** on the drawing area.
   The first **Up** entry box is selected.
5. Click a location on the positive **y-axis** on the drawing area.
6. Click the **OK** button.
   The R**edefine Plane** dialog box is closed and the new plane is defined and added to the **Planes** submenu in the **3D** menu. You can rename the new plane named **Plane 1** using the **Rename** button in the **Define Plane** dialog box.

Redefining a plane

Two different methods exist for redefining a plane:

By a temporary plane

1. Define a *temporary plane* using the **3-Point Plane** command.
2. Select the **Define Plane** command in the **3D** menu.
   The **Define Plane** dialog box is displayed listing the *temporary plane* in the **Planes** list box.
3. Delete the plane of which you want to change the orientation.
4. Select the **TempPlane** in the **Planes** list box and rename the plane with the name of the deleted plane in the **Rename** entry box.
5. Click the **Rename** button.
5. Click the **OK** button.
   The plane is redefined and the **Define Plane** dialog box is closed.

You can modify the *Origin* and the *Orientation* of any plane except the **World** (*Top*) Plane.

By definition

1. Select the **Define Plane** command in the **3D** menu.
   The **Define Plane** dialog box is displayed.
2. Select the plane of which you want to change the orientation.
   If necessary, click the *Lock* icon to unlock the plane.
3. Click the **Redefine** button.
   The **Redefine Plane** dialog box displays and the **Origin** specification is already highlighted.
4. On the drawing area, click the location for the new origin.
   The first **Right** entry box is selected.
5. Click a location on the positive **x-axis** on the drawing area.
   The first **Up** entry box is selected.
6. Click a location on the positive **y-axis** on the drawing area.
7. Click the **OK** button.
   The **Define Plane** dialog box is closed and the plane is redefined as indicated.

You cannot redefine the standard work planes (**Front** and **Right**) without unlocking them. You can never redefine the **World** (*Top*) plane, which was established from the original orientation of the model space.

Set Plane to Screen

This command sets the work plane to be coincident with the screen, where the **Origin** of the plane is in the *Center* of the screen; the ***x****-axis* is coincident with the *Width* of the screen and the ***y****-axis* is coincident with the *Height* of the screen. This sets the *Work Plane* to match the screen in all views which is particularly useful when you are moving from view to view while drafting.

The **Set Plane to Screen** command creates a *temporary plane* that you can name using the **Define Plane** command. The **Set Plane to Screen** command sets not only the plane in the active view, but in all existing views (including all *Detail Views*) parallel to the screen.

The Design Process

In ***DraftBoard*** you can create three-dimensional wireframe models. You can view them from different angles and export them as wireframe models. For the creation of wireframe models you proceed as follows:

- You create the three-dimensional wireframe model. With the help of the *Drawing Assistant* you can design this model as easy as isometric drawing on a drafting board. We design the model in a three-dimensional work space using one or several view windows, to observe multiple view orientations while we are modelling.
- Optionally you can easily derive from the 3D model two-dimensional draftings for manufacturing containing several views on a drawing format with dimensions and notations.

First we will look at the additional tools and features that are specific to 3D.

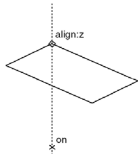
3D Features and Tools

All of the tools you would use to create 2D geometry in ***DraftBoard*** would also be used to create 3D geometry. However, there are some tools and features that are exclusive to 3D. One of the most important features in 3D again is the *Drawing Assistant*.

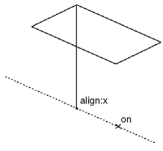
3D Drawing Assistant

The *Drawing Assistant* is unique to ***DraftBoard*** and makes ***DraftBoard*** easy to use in 3D because it thinks like a designer. It displays temporary construction lines, provides information about existing geometry, and displays notations of the relationship between new and existing geometry and it does this in three dimensions. The following are examples of the alignment notations for ***x***, ***y***, and ***z*** *axes*:

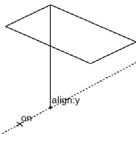
align: z A dynamic construction line along the ***z*** *axis*, perpendicular to the work plane.



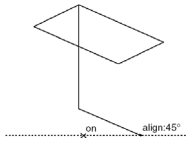
align: x A dynamic construction line along the ***x*** *axis*, parallel to the work plane.



align: y A dynamic construction line along the ***y*** *axis*, parallel to the work plane.

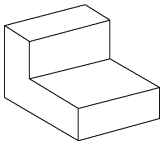


align: 45° A dynamic construction line along the ***45°*** *angle*, parallel to the work plane.



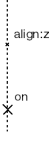
Isometric Drawing with the Drawing Assistant

With the help of the *Drawing Assistant* 3D design becomes as easy as isometric drawing on a draftboard. The only thing we have to change is the view orientation so that we can see all three directions at once. In the following example we will choose a *trimetric view*:

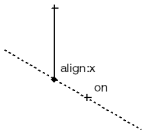


1. Choose **Trimetric** from the **View Mode** submenu in the **Views** menu.

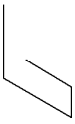
   The view and with it all temporary construction lines of the *Drawing Assistant* will be aligned trimetrically.
2. Select the **Connected Line** tool from the **Line** subpalette on the *Tool palette*.
3. Begin by clicking in the middle of the drawing area and move the pointer straight down along the ***align: z*** construction line about **2 cm** and click.



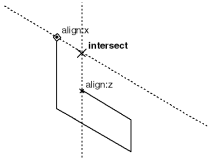
4. Move the pointer along the ***align: x*** construction line about **2 cm** and click.



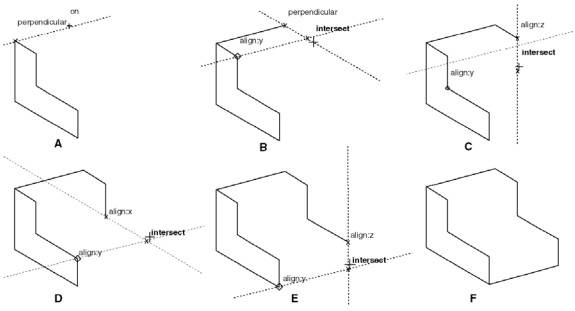
5. Continue clicking connected lines until your drawing appears as below:



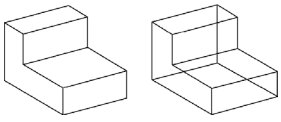
6. Move the pointer until the ***intersection*** notation appears and click.



7. Continue clicking according to the following pictures.



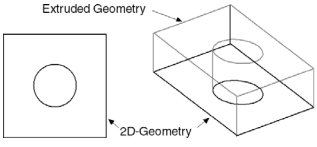
8. Draw the three unconnected line with the **Single Line**.



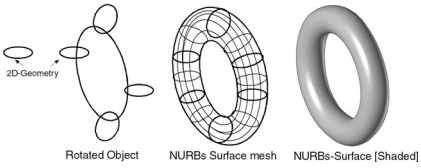
In this small example you could see that 3D design with ***DraftBoard*** is as easy as isometric drawing only faster and more precise.

Extruding and Revolving of Objects

The two commands **Extrude** and **Revolve** in the **3D** menu, allow you to create a 3D model from 2D geometry. The **Extrude** command adds depth to a 2D object, while the **Revolve** command copies and rotates a 2D object to provide three dimensions.



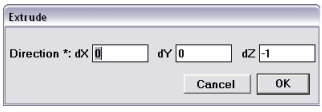
A 2D part is extruded to a 3D model.



A circle is rotated with the **Revolve** command to create a torus.

Extrude

This command adds depth to a 2D object. When you select the **Extrude** command from the **3D** menu the following dialog box is displayed:



The **Extrude** dialog box contains the following elements:

Direction In the entry fields **dx**, **dy** and **dz** you specify the number of units for the *Height* of the 2D object and the *Direction* (**x**, **y**, or **z** axis) in which the 2D object is to be extruded.

You can enter either positive or negative numbers to indicate the *units* and *direction*, or you can ***drag*** the mouse to specify the distance and direction of the extrusion.

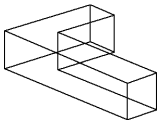
Extrude Surface When you select this option the 2D geometry is surfaced the same time is extruded.

As default the **x** and **y** are on the current work plane and **z** is perpendicular to that plane. In the **Top** view the positive **z** axis is extending toward you and the negative **z** axis away from you with the screen as origin. This is not necessary relative to the original ***x***, ***y***, and ***z*** *axis* that were used when the geometry was constructed. Therefore you should always align the *current work plane* with that side of the object you want to be extruded.

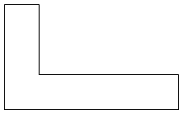
Important*: It is possible to indicate the extrusion values by* ***dragging*** *a vector on the drawing sheet in an* ***Isometric*** *or* ***Trimetric*** *view. Therefore you should always switch to one of these views before extruding any 2D object. Don't forget that the extrusion angle between the extrusion vector and the current work plane must be always greater* ***0****.*

Extruding objects

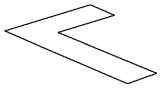
To create the following bracket by extruding a basic 2D shape you have to proceed as follows:



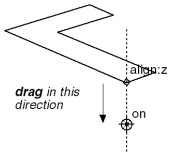
1. Select the **Top** view from the **Views** submenu in the **Views** menu.
2. Draw the side of the bracket using the **Connected Line** tool.



3. Select the geometry if it is not already selected.
4. Select **Trimetric** from the **Views** submenu in the **Views** menu.
   The side view of the bracket is aligned *trimetrically*.



5. Choose the **Extrude** command from the **3D** menu.
   The **Extrude** dialog box is displayed.
6. Click in one of the direction entry fields and ***drag*** then with the mouse a vector along the ***z*** *axis* in the drawing area to indicate the *direction* and *distance* of the extrusion.

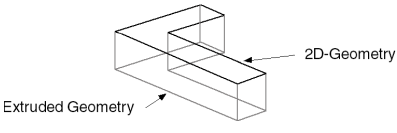


The values ***dragged*** for the *direction* and the *distance* of the extrusion are automatically entered into the **Direction** boxes.

All values you enter, use the current units set in the **Preferences** submenu of the **Layout** menu.

7. Overwrite the length of the vector ***dragged*** in the **dx** box with the desired height of the bracket. (Positive and negative values are allowed.)
8. Click the **OK** button.
   The object is extruded.

When you have selected the option **Extrude Surfaces** the object is extruded and surfaced the same time. See as well the section ***NURB*** *Surfaces* in the next chapter

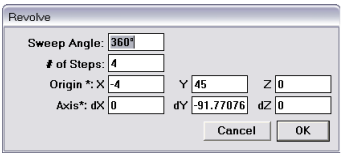


Tip: *Before you extrude any 2D geometry you should consider which face of the object shows the most detail and whether the extrusion will be uniform or not if some portions of the part do have different dimensions. Depending on these considerations you should draw the 2D geometry.*

Revolve

The **Revolve** command copies and revolves a 2D object into a 3D object around a specified revolution axis.

When you select the **Revolve** command from the **3D** menu the following dialog box is displayed:



The **Revolve** dialog box contains the following elements:

Sweep Angle In this field you enter the number of degrees (**Sweep Angle**) for the revolution.

# of Steps In this field you enter the number of copies or divisions.

Origin* In these entry fields you enter the coordinates for the *Startpoint* of the *Revolution axis*.

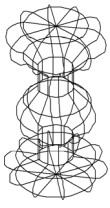
Axis* In these entry fields you enter the coordinates for the *Endpoint* of the *Revolution axis*.

Revolve Surface When you select this option the 2D geometry is surfaced the same time is revolved.

You indicate the *Axis of Revolution* also by double clicking the **Origin** box and ***dragging*** a vector for the direction of the *Revolution axis* in the drawing area. The length of the vector is unimportant. The values of the ***dragged*** vector are automatically entered into the **Origin** and **Axis** entry fields.

Revolving objects

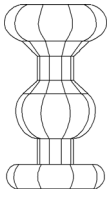
To revolve a 2D object into a 3D object you begin by drawing half of the object on an axis for revolving.



1. Draw the geometry to be revolved as in the following graphic.

2. Select the geometry if it is not already selected.
3. Choose the **Revolve** command from the **3D** menu.
   The **Revolve** dialog box is displayed.
4. Specify the **Sweep Angle** (360°) for the revolution.
5. Specify the number of copies (**2**) in the **# of Steps** entry field.
6. Click the **Origin** (*Startpoint*) of the *revolution axis*.
7. Click the other end (*Endpoint*) of the *revolution axis*.

Important: *Normally the **# of Steps** should be **2**. Only when the **Sweep Angle** is **360°** you should enter **3** or **4** for the **# of Steps**.*

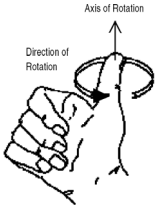


You indicate the *Axis of Revolution* also by double clicking the **Origin** box and ***dragging*** a vector for the direction of the *Revolution axis* in the drawing area.

The selected geometry is now revolved around the axis as specified, according to the right hand rule of revolution.

Right-Hand Rule of Revolution

The direction of revolution is determined by the *Right-Hand Rule of Revolution* which states: if that the thumb is pointed toward the positive *Axis of Revolution*, the revolution will be in the same direction in which the fingers are curled.



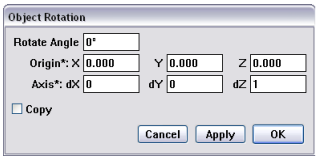
Rotating Objects

In ***DraftBoard*** you can not only rotate the *view orientation* to view the geometry from any angle, but also objects in relation to other objects.

Object Rotation

The **Object Rotation** command in the **3D** menu allows you to rotate geometry around any axis you specify.

The **Object Rotation** command displays the following dialog box:



The **Object Rotation** dialog box contains the following elements:

Rotate Angle In this field you specify the *angle* you want to rotate the geometry.

You indicate the *Axis of Rotation* also by double clicking the **Origin** box and ***dragging*** a vector for the direction of the *Rotation axis* in the drawing area. The values of the ***dragged*** vector are automatically entered into the **Origin** and **Axis** entry fields.

Origin* In these entry fields you enter the coordinates for the *Startpoint* of the *Rotation axis*.

Axis* In these entry fields you enter the coordinates for the *Endpoint* of the *Rotation axis*.

Copy When you select this option a copy of the geometry will be rotated.

Apply When you click the **Apply** button the geometry will be rotated as specified and you can select another object for next object rotation.

Rotating objects

With the **Object Rotation** command you rotate geometry in relation to other objects.

1. Select the geometry you want to rotate..
2. Choose the **Object Rotation** command from the 3**D** menu.
   The **Object Rotation** dialog box is displayed.
3. Enter the angle you want to rotate the geometry into the **Rotate Angle** data field.
4. Click the **Origin** (*Startpoint*) of the *rotation axis*.
5. Click the other end (*Endpoint*) of the *rotation axis*.
6. Select the **Copy** option, if you want to rotate a copy of the selected object.
7. Click the **Apply** button.
   The object is rotated around the specified axis.
8. Select the next object for rotation or close the **Object Rotation** dialog box.

You also can indicate the *Axis of Rotation* by double clicking the **Origin** box and ***dragging*** a vector for the direction of the *Rotation axis* in the drawing area.

3D Construction Rules

For many of the ***DraftBoard*** *Drawing Tools* and *Commands* certain rules must be observed

Drawing Tools in 3D

All ***DraftBoard*** *Drawing tools* behave in a three-dimensional work space according to certain rules. When you know these rules you can create any object geometry using these tools.

The **Tracer** tool does not work with 3D objects.

Lines	**Single**, **Connected** and **Parallel Lines** have no 3D restrictions.
	Tangent lines pulled off a curve, circle or spline will be tangent if created in the same plane as the curve, circle or spline.
Arcs	**Center-Point Arcs** will be drawn parallel to the work plane.
	3-Point Arcs and **Tangent-Point Arcs** will be in the plane designated by the three points defining the arc.
Circles	**Center-Point Circles** and **Opposite-Point Circles** will be drawn parallel to the work plane.
	3-Point Circles will be drawn in the plane designated by the three points defining the circle.
	Tangent Circles will be in the plane of the two defining entities.
Ellipses	**2-Point Center Ellipses** and **Opposite-Point Ellipses** will be drawn parallel to the work plane.
	3-Point-Center Ellipses and **3-Corner Ellipses** will be drawn in the plane designated by the three points defining the ellipse.
Rectangles	All *rectangles* and *polygons* will be created parallel to the work plane.
Splines	*Splines* have no 3D restrictions.
Fillets and Chamfers	*Fillets* and *Chamfers* are constructed in the plane defined by the selected entities.

Tangent lines tangent only to one object can be created only with objects on the work plane at the origin (See as well *Specifying the position of a work plane* in this chapter). Tangent lines tangent to two objects can be created on any work plane as long as both objects are on the same work plane.

Editing Objects in 3D

The **Simple Trim** tool, the **Relimit** tool and the **Segment** tool work independent from the current work plane as in 2D.

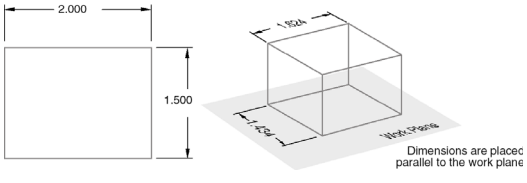
The **Transformation** tools behave as follows:

Move	Object geometry can be moved in any direction.
Rotate	Selected geometry can be rotated around a specified point parallel to the ***z*** *axis* of the current work plane.
Expand/Shrink	Object geometry can be scaled in any direction.
Stretch	Object geometry can be stretched in any direction.
Mirror	Selected geometry can be mirrored parallel to the current work plane along a specified *mirror axis*.

Before you rotate an object it is recommended to define a plane with the **3-Point Plane** command perpendicular to the plane where the rotation point is located.

Text and Dimensions

Text and *Dimensions* will be always created on the current work plane of the active view. Therefore you should set the work plane to correspond to the view before you begin adding *text* or *dimensions* in a view.



Another elegant method for dimensioning 3D objects is to align the face of the object you want to dimension parallel to the screen and then to use the **Set Plane to Screen** command.

Dimensioning in 3D

All linear dimensions are placed parallel to the current work plane. All other dimensions behave as follows:

Point to Point	*Point to Point dimensions* are placed parallel to the current work plane.
Perpendicular	*Perpendicular dimensions* are created in the plane of the baseline.
Radial and Diametral	*Radial* and *Diametral dimensions* are created in the plane of the arc, circle or fillet.
Anglular	*Angular dimensions* are created in the plane of the two lines whose angle is measured.
GD&T	*GD&T dimensions*, *Welding* and *Surface symbols* are placed parallel to the current work plane.

Hidden Lines

When you want to dimension surfaced objects in the **Hidden Lines** view mode you have to consider the following:

Even though only the visible edges if the geometry are displayed in the **Hidden Lines** view mode, the original wireframe geometry and the surface mesh of the ***NURB*** *Surfaces* are still existing in the background.

While dimensioning the *Drawing Assistant* snaps to the *construction points* of the original wireframe geometry that is lying invisible behind the displayed object edges. The *Drawing Assistant* can't identify the visible objects edges since they are not made up by *single lines* but by so called *segmented polylines* whose construction points the *Drawing Assistant* is unable to snap to.

This is also valid for the **Flatten View** command, when used for a view displaying a surfaced object in the **Hidden Lines** view mode, because this command flattens not only the visible object edges but also the invisible original wireframe geometry.

In this case the visible object edges are placed onto the current layer and the original wireframe geometry goes onto the layer **Wireframe**.

The layer **Wireframe** is automatically created and hidden with the **AutoSurface** command and must be redisplayed when dimensioning circles in a flatten view, since the visible edges cannot be dimensioned.

Hatches and Solid Fills

The **Tracing** tool does not work with three-dimensional objects.

Hatches and *Fills* are placed parallel to the current work plane only in that view in which they were crated. In other words hatches and fills will be only displayed correctly when you can look at the hatch or fill along the ***z*** *axis* of the plane.

Parametrics in 3D

The parametric feature works in 3D with some restrictions. You might consider its functionality as **2½ D** since it functions properly in all planes parallel to the current work plane.

The parametric mechanism has two phase:

1. In the first phase, parametrics ignore the ***z*** *coordinates* of the selected geometry and resolves the geometry as it does in 2D. Any dimension that is not wholly in a plane parallel to ***x, y*** *plane* is also ignored. In other words, the ***x*** and ***y*** *coordinates* are resolved and the ***z*** *coordinates* are not changed.

Be aware that if you get errors back, the **highlighted lines** and **points** are drawn right on the work plane, even though the geometry may not be drawn on the work plane.

2. In the second phase, parametrics adjust the ***z*** *coordinates*, as necessary. All linear dimensions parallel to the ***z*** *axis* are examined and every point in the selected geometry is considered to define a plane parallel to ***x***, ***y*** *plane*. The dimensions in the ***z*** *direction* define the required distance between the planes. Unlike the first phase, in the second phase parametrics is very tolerant of missing dimensions. If the dimensions do not completely define the distance between any two planes, then the distance is not changed.

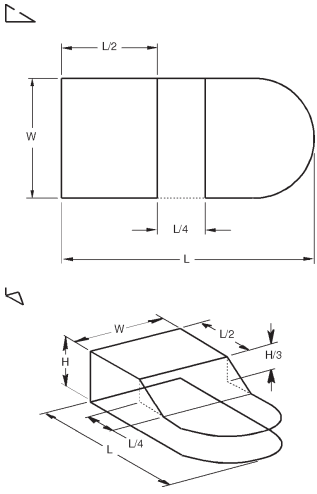
In **Phase II** any line or arc that does not lie in a plane parallel to the ***x***, ***y*** *plane* is modified appropriately because the defining points are adjusted by parametrics and the line or arc is changed accordingly.

This includes all lines parallel to the ***z*** *axis* and any oblique lines at any angle to all three axes.

Note that parametrics deals with all geometry in the current work plane coordinates. Consequently, it is a good idea to set up a view that looks down the ***z*** *axis* of the work plane onto the ***x***, ***y*** *plane*. In this way, you can see what the first phase of the operation „sees". This is particularly useful if you encounter problems resolving parametric geometry.

Errors in parametrics are shown in **bold lines** drawn in the ***x***, ***y*** *plane* and independent of the ***z*** *values* of the highlighted lines. The easiest way to see this is to look down the ***x***, ***y*** *plane* by choosing the **View the Plane** command from the **3D** menu.

In the following drawing the first phase of parametrics will deal with only four dimensions **W**, **L**, **L/2** and **L/4**. In the second phase the **H** and **H/3** dimensions are evaluated.



The **L/4** dimension was created with the **Horizontal Dimension** tool, even though the dimensioned points have different ***z*** *values*. The **H** and **H/3** dimensions were created with the **Vertical Dimension** tool while the work plane was set to the ***y***, ***z*** *plane*.

CHAPTER

17

3D Surfaces

Construction Rules

Calculating Surfaces

Visualization

Exporting Surfaces

3D Surfaces

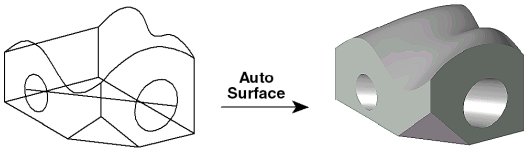
In ***DraftBoard*** you create three-dimensional wireframe geometry that you can *surface*, view in *hidden line* or *shaded* view mode and you can *illuminate* using different light surface.

When creating ***NURB*** *Surfaces* there are a number of things you need to understand about how the *Autosurfacing* routine works. You need to know what to expect and you will need to pay a little bit more attention to the detail of how you construct your 3D wireframe models.

IMPORTANT! = Not all 3D Features (such as **AutoSurface**) are in all country versions and/or for all operating systems available. Check with your local distributor.

Construction Rules

The automatic calculation of ***NURB*** *Surfaces* is done with the **Auto Surface** command in the **3D** menu.



For the successful creation of ***NURB*** *Surfaces* you must observe some rules while constructing 3D wireframe geometry you want to *autosurface*.

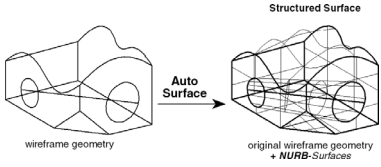
Surface Types

The **Auto Surface** mode distinguishes between several type of surfaces:

Structured Surfaces

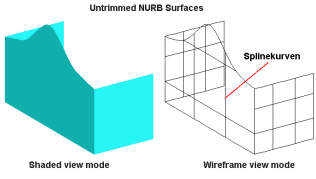
The **Auto Surface** command always creates ***Structured Surfaces***. This is a grouped entity composed of the calculated ***NURB*** *Surfaces* (displayed as a *wireframe mesh*) and the original wireframe geometry.

In other words a ***Structured Surface*** consists always of the *original wireframe geometry* plus the calculated ***NURB*** *Surfaces*.



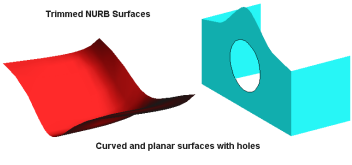
NURB Surfaces

Auto Surface creates always ***NURB*** *Surfaces*, that are defined by ***NURB*** *Splines*, even they are *planar surfaces*.



Trimmed NURB Surfaces

Surfaces are called *trimmed* ***NURB*** *Surfaces* whenever they are curved or their boundaries are trimmed (for example by a hole).

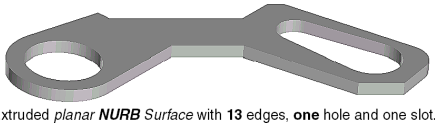


Detailed Construction Rules for Surface Models

A summary of these rules you find at the end of this section.

1. ***Planar*** *Surfaces* with any number of edges are supported with or without holes in the face.

Holes can be only calculated for three-dimensional objects. An isolated surface with a hole cannot be calculated.

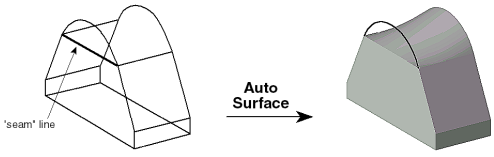


Extruded *planar* ***NURB*** *Surface* with **13** edges, **one** hole and one slot.

The **Autosurfacing** routine works successfully with any *Planar Surface*, no matter how many edges it has, and if there are holes in it or not.

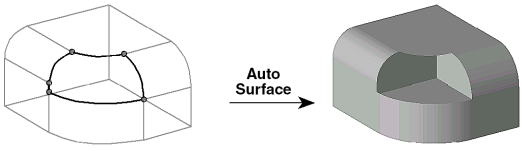
2. ***Planar*** *Surfaces* should have **no** geometry that would act as a "seam" running across the face, but only geometry that either represents a boundary or a hole.

   All geometry that connects across a face one boundary object to another, or a boundary object to a hole object, or runs across a hole can cause problems.



The front "seam" line causes a this autosurface problem.

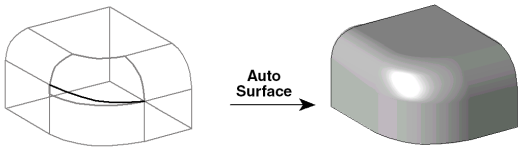
3. ***Non-Planar*** *Surfaces* are not allowed no have more than ***4*** *edges*.



The **highlighted** corner face has **5** edges and gets therefore ignored.

Any ***Non-planar*** *Surface* that has more than ***4*** *edges* must be broken up into two separate surfaces and each one must have four or fewer edges. This is done by adding more wireframe geometry.

In the following example the ***Non-Planar*** *Surface* is broken up into two surfaces with a maximum of ***4*** *edges* by adding an *ellipse*.



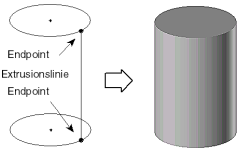
An additional Ellipse brakes the surface into two separate surfaces with **4** edges or less.

4. ***Non-planar*** *Surfaces* are not supported.

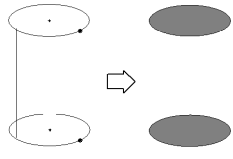
   This is largely because in most cases it is not possible to represent the hole on a non-planar surface with any wireframe geometry.

If you don't construct the cylinder step by step with lines and circles, but use the **Extrusion** command the *Extrusion Line* will be automatically connected with the ***endpoints***.

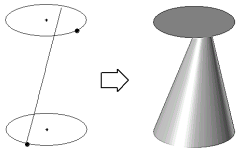
5. When ***extruding*** circles to make cylinders, the extrusion line **must** be connected to the ***Endpoint*** (*vertex*) of both circles.



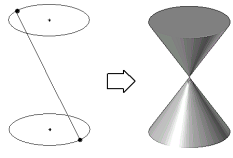
Extrusion Line connected through ***endpoints***.



Extrusion Line not connected through ***Endpoints***.



One endpoint **on**, one endpoint **off**.



Endpoints rotated.

6. Don't have more than <u>one</u> extrusion line for circles that have been extruded into cylinders.

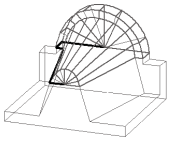
   When extruding a circle into a cylinder to create a wireframe model, it makes sense to make 4 or 8 copies of the extrusion line around the cylinder in order to make it look more like a cylinder. For surfaced models simplicity is the key for successful autosurfacing. More than one extrusion lines isn't necessary, and might even cause problems.

7. When doing a **Revolve** operation, use **1** or **2** for **# of Steps**.

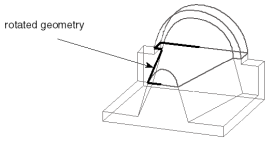
   A higher number of "spoke" lines aren't necessary and may cause trouble.

   Only for ***360°*** *Revolutions* the **# of Steps** should be **3** or **4**.

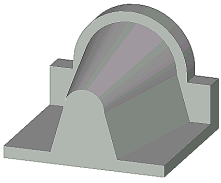
   The fastest way to create a sphere is to revolve a circle by **90°**. The ***Revolution*** *axis* should run through the ***endpoint*** (vertex) of the circle not through one of the ***quadrant*** points.



Spoke Lines with **# of steps** set to **9.**



Clean faces with **# of steps** set to **2.**



8. You must always connect wireframe geometry ***endpoint*** to ***endpoint***.

   Don't have *overlapping* lines, or *duplicate* lines on top of each other. Even though ***Autosurfacing*** *routine* handles these extra lines quite tolerant, you can avoid many problems by creating clear wireframe geometry.

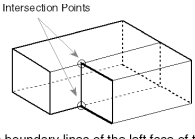
   The only exception to this is the next rule.

9. When the boundary line of one surface is intersected somewhere along its length by the boundary line of a second surface, ***DraftBoard*** will divide each boundary line into two separate line pieces and will build the surfaces based on that new geometry.

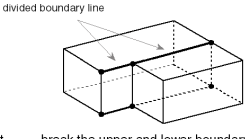
   The surface calculation will be only successful, if boundary lines are not broken into more than <u>two</u> pieces (separate line segments).

   If a boundary line is intersected by two lines of another surface such that the first boundary line would be broken into three pieces, the ***Autosurfacing*** *routine* will fail.

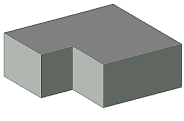
   This automatic divisions only occur with ungrouped objects. If one of the boundary lines is part of an object that is **grouped** then no division will occur.



The boundary lines of the left face of the front cube intersect the front face of the rear cube and ...

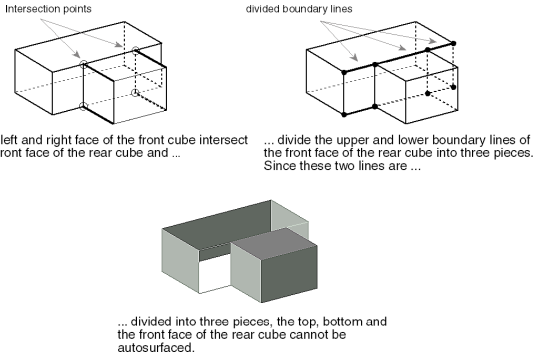


... break the upper and lower boundary lines of the front face of the rear cube into two pieces. Since the these two lines are ...



... only divided into two pieces, all surfaces can be autosurfaced.

In the following example the front face of the back cube is broken into <u>three</u> pieces.


Intersection points
divided boundary lines
The left and right face of the front cube intersect the front face of the rear cube and ...
... divide the upper and lower boundary lines of the front face of the rear cube into three pieces. Since these two lines are ...
... divided into three pieces, the top, bottom and the front face of the rear cube cannot be autosurfaced.

10. Whenever you want to delete the surfaces and return to the original model try to use **Undo** instead of **Ungroup**.

    When you ungroup a surfaced model using the **Ungroup** command and delete its surfaces, you may get back a different wireframe model then the one you started with. Because of *Rule* ***9***, some lines may be divided that weren't divided before.

11. When working with multiple objects, select each one individually and autosurface them one at a time for best results.

Summary of Rules

Surfaces are called ***planar*** *surfaces* when all *surface points* lie in a two-dimensional plane.

1. ***Planar*** *surfaces* may have *any number of edges* and *any number of holes* in the face.
2. ***Planar*** *surfaces* should have no geometry that would act as a "seam" running across the face, but only geometry that either represents a boundary or hole.
3. ***Non-planar*** *surfaces* are not allowed to have more than ***4*** *edges*.
4. ***Non-planar*** *surfaces* are not allowed to have any *holes*.

When a circle is extruded using the **Extrude** command this happens automatically.

5. When ***extruding*** circles to make cylinders, the extrusion line must be connected to the ***endpoints*** (*vertex*) of both circles.
6. *Circles* extruded into *cylinders* are not allowed to have more than one extrusion line.
7. When doing a **Revolve** operation, use **1** or **2** for **# of Steps**. Only for ***360°*** *Revolutions* the **# of Steps** should be **3** or **4**.

   If revolving a circle, the **Axis of Revolution** should run through an ***endpoint (vertex)***, the **Revolution Angle** should be **90°** and the **# of Steps** should be **2**.
8. Wireframe geometry must be always connected ***endpoint*** to ***endpoint***. Don't have *overlapping* or *duplicate lines* on top of each other.

   The only exception to this is the next rule:
9. Boundary lines of surfaces may be divided by the boundary line of a second surface into two separate line segments at most

   If a boundary line is broken into three pieces, the ***Auto-surfacing*** *routine* will fail.

   Automatic divisions occur only with *ungrouped* objects.
10. Whenever you want to return to the original wireframe geometry after having *autosurfaced* a part, use the **Undo** command instead of the **Ungroup** command.
11. When working with multiple objects, select each one individually and *autosurface* them one at a time for best results.

Calculating Surfaces

When you have constructed a wireframe model according to the rules described in the section above, you can surface the part automatically using the **Auto Surface** command in the **3D** menu.

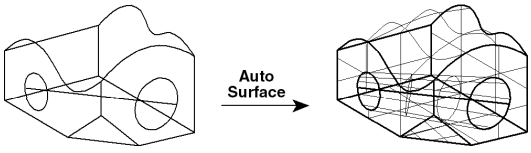
The ***Autosurface*** *feature* is a unique technology of ***DraftBoard*** that you don't find in any other ***CAD*** *software*. With a simple mouse click ***DraftBoard*** takes your wireframe geometry and surfaces it.

Auto Surface

The **AutoSurface** command in the **3D** menu calculates automatically for a *wireframe model* a surfaced object, that can be exported as a ***Surface*** or ***Volume*** *model*.

You only must select the wireframe geometry and select then the **AutoSurface** command from the **3D** menu. It doesn't matter if the wireframe geometry is selected on the *drawing sheet* or in a *detail view*.

When the surface calculation is finished the resulting surfaces are represented by a *wireframe surface mesh*. The structured surfaces are grouped an can be edited as one single object.



The ***Structured Surface*** consists of the *original wireframe geometry* plus the calculated ***NURB*** *Surfaces*.

If you ungroup any structured surface it is dissolved into the ***NURB*** *Surfaces* and the *wireframe geometry* (See as well *Editing Structured Surfaces* in this chapter).

Structured *Surfaces* are always placed onto the current layer regardless of the layer on which the original wireframe geometry is located.

Calculating surfaces with the Auto Surface command

1. Construct the 3D wireframe geometry to be autosurfaced considering the respective rules.
2. Select the geometry you want to autosurface.
3. Select from the **Color** submenu in the **Pen** menu the desired color for your surfaced geometry.

   For a good representation of the shaded model any color except black is recommended.

   Auto Surface represents calculates surfaces using the *current pen color* specified in the **Color** submenu in the **Pen** menu.
4. Choose the **Auto Surface** command from the **3D** menu.

   DraftBoard identifies the surface boundaries for the selected geometry and calculates the respective surfaces. A *wireframe surface mesh* appears on the screen that represents the calculated surfaces.

   The calculation time is depending on the *size* and *complexity* of the selected geometry and the *available memory* of your computer.

   If the **Auto Surface** routine has trouble interpreting the selected geometry an error box will appear and the problematic geometry is highlighted. (See as well ***Auto Surface*** *Error Messages* in this chapter.)

   When you acknowledge the error message only the recognized surfaces are represented. Unrecognized boundary lines are shown in the original geometry color.

   In this case you must return to the *original wireframe geometry* using the **Undo** command in the **Edit** menu to correct the problem.

   See as well *Editing Structured Surfaces* in this chapter.

For more information about displaying surfaced geometry in **Hidden Lines** or **Shaded** view mode see *Visualizing Objects* in this chapter.

Creating a non-associative Surface Model

In case you want to edit the wireframe geometry later independent from the surfaced object, it is recommended to use a copy of the wireframe geometry for the surface calculation. For that you proceed as follows:

1. Construct the wireframe geometry to be autosurfaced considering the respective rules.
2. Create a *Detail View window* using the **Detail View** tool.
3. Activate the detail view and select the **Model** command in the **Views** menu.

4. Create a new model and make it the active model for the detail view.
5. Select the wireframe geometry to be autosurfaced on the sheet.
6. Choose the **Copy** command in the **Edit** menu.
7. Activate the detail view and choose the **Paste** command in the **Edit** menu.

   The wireframe geometry is copied as a *non-associative* model into the detail view.
8. Select the wireframe geometry in the detail view.
9. Choose an appropriate color from the *Color Palette* in the **Pen** menu.
10. Select the **Auto Surface** command in the **3D** menu.

    The part is autosurfaced.
11. Add *dimensions* and *text* to the wireframe geometry on the sheet.

Naturally it is also possible to dimension a surfaced object in the **Shaded** or **Hidden Line** view mode. See as well *Visualizing Objects* in this chapter.

Surfacing complex objects

For surfacing complex objects it is recommended to divide the geometry into several objects and place these objects onto separate layers before surfacing each object individually.

While surfacing one object all other objects (including surfaced objects) should be hidden using the **Layer** command.

This way the calculation time is reduced as well as the time required to refresh the screen. In addition this method allows to shade each object in a different color. (See as well *Visualizing Objects* in this chapter.)

Deleting Surfaces

Deleting calculated surfaces restores the original wireframe geometry and has the same effect as using the **Undo** command in the **Edit** menu.

1. Select the surfaced geometry.
2. Choose the **Ungroup** command in the **Arrange** menu.

   A dialog box is displayed asking you if you want to keep or delete the surfaces.
3. Click on the **Delete Surfaces** button.

   All surfaces will be deleted and the original wireframe geometry will be restored except all segmented boundary lines that remain divided.

Therefore you should always use he **Undo** command to return to the original wireframe geometry.

Editing Surfaces

1. Select the surfaced geometry.
2. Choose the **Ungroup** command in the **Arrange** menu.

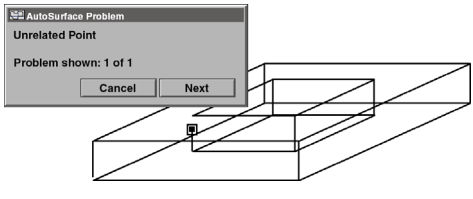
   A dialog box is displayed asking you if you want to keep or delete the surfaces.
3. Click on the **Keep Surfaces** button.

   The grouped ***Structured Surface*** composed of the ***NURB Surfaces*** and the *original wireframe geometry* will be ungrouped into *Trimmed NURB Surfaces* and the underlying *wireframe geometry*. Now you can select each trimmed surface, move it, change its color or delete it individually.

When you group the ungrouped surfaces again you will not return to the previous ***Structured Surface***. To restore the ***Structured Surface*** you have to use the **Undo** command in the **Edit** menu.

Auto Surface Error Locator

If ***DraftBoard*** encounters unconnected geometry that will prevent any wireframe geometry from properly surfacing, an error box will appear, similar to that displayed for *parametrics*. It will state the *type* and *number* of problems.



Simultaneously the identified point causing the problem in the wireframe geometry is highlighted.

If there is more than one problem indicated, click the **Next** button to display it.

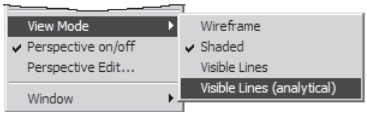
Take note of the relationships between the problematic geometry. You must return to your original wireframe to correct your geometry using the **Undo** command in the **Edit** menu..

Visualizing Objects

Surfaced models can be *illuminated*, displayed in a *perspective view* or in a *hidden lines* or *shaded* view mode.

View Modes

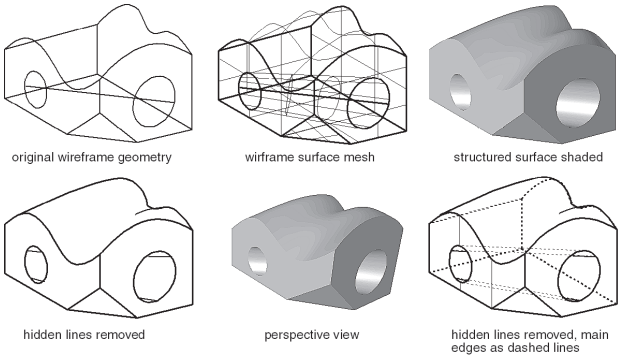
All commands for the different view modes are located in the **View Mode** submenu in the **Views** menu.



All view modes can be used for both the *Sheet View* and for any *Detail View window*.

The **View Mode** submenu in the **Views** menu contains the following commands: *Wireframe*, *Shaded*, *Visible Lines* and *Visible Lines (analytical)*. **Wireframe** is the default view mode.

For the **Shaded** and the **Hidden Lines** mode it is possible to activate a *Perspective view*. For the **Hidden Lines (analytical)** mode you can specify if the hidden lines should be represented by *solid* or *dotted* lines.



Changing the View Mode

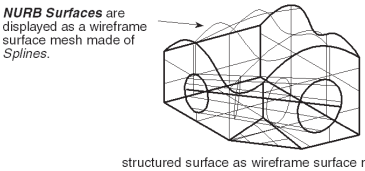
The view mode can be changed any time using the view mode commands in the **View Mode** submenu in the **Views** menu, but they will affect only the representation of surfaced objects. Changing a view mode applies automatically for all surfaced objects in the active view (*Sheet* or *Detail View*).

Editing Structured Surfaces

Surfaced objects cannot be edited in any of the view modes. You first must return to the original wireframe geometry using the **Undo** command in the **Edit** menu or the **Ungroup** command in the **Arrange** menu, before you can edit any surfaced object.

Wireframe

This view mode shows the *structured surface* as a *wireframe surface mesh* representing the calculated ***NURB*** *splines*.



structured surface as wireframe surface mesh

You can rotate surfaced objects in any view mode using the *Trackball*. ***Draft-Board*** will automatically recalculate and redisplay the image in the new view orientation.

Shaded

This mode displays surfaced geometry as a solid looking object. The object must not be selected for a *shaded* representation.

Due to the new ***OpenGL*** *Support* 3D models can in real time be rotated and to a certain extend edited in the *Shaded View Mode* (*move*, *copy* etc.). Also commands such as **Polar** and **Linear Duplicate** can be executed very fast.

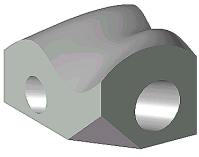
The display speed (depending from the processor used and the available memory) can be accelerated further by ***OpenGL*** *Graphic* boards.

Further settings for *View precision*, *dynamical Rotation* etc. are done in the **Visualization** dialog window in the **Preferences** menu of the **Layout** menu.

Displaying surfaced geometry in the Shaded view mode

For a good representation of the shaded model any color except *black* is recommended.

1. Select any autosurfaced geometry.
2. Select a new color in the **Color** submenu in the **Pen** menu, if you want the object to appear in a color different from the current pen color.
3. Click on the drawing area to deselect the geometry.
4. Choose **Shaded** from the **View Mode** submenu in the **Views** menu.



Flächenverbund schattiert

The shaded geometry is displayed with the default light settings using the default grey background color. How to change the background color and the light settings is described in the *Preferences* and *Light Setting* sections of this chapter.

Shading geometry in different colors

It is possible to shade surfaced objects in different colors by autosurfacing each differently colored object individually.

When you want to shade an already surfaced model in different colors you must use the **Ungroup** command, keep the surfaces and then assign different colors to the individual parts.

1. Crate all wireframe objects to be autosurfaced.
2. Assign to each geometry the desired color from the **Color** submenu in the **Pen** menu.
3. Autosurface each colored part one after another.
4. Choose **Shaded** from the **Views** submenu in the **Views** menu.

   All object parts are shaded in the specified color.

Hidden Lines

These modes display surfaced geometry such that only the front most lines are visible. The object must not be selected for a *Hidden Lines* representation. There are two view modes available for hidden lines:

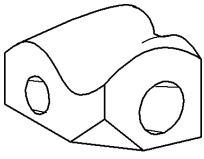
- Visible Lines

  and
- Visible Lines (analytical).

Visible Lines

This view mode shows all visible edges of a model and hides all model edges that are covered by surfaces.

You can rotate surfaced objects in any view mode using the *Trackball*. ***DraftBoard*** will automatically recalculate and redisplay the image in the new view orientation.



View mode **Hidden Lines**

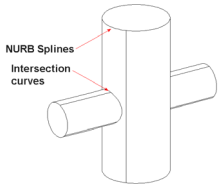
This view mode is also supported by ***OpenGL*** and represents models as pixel graphics.

Further settings for *View precision*, *dynamical Rotation* etc. are done in the **Visualization** dialog window in the **Preferences** menu of the **Layout** menu.

How you can display the *hidden lines* using solid or dotted lines is described in the next section.

Visible Lines (analytical)

This view mode shows all visible edges of a model and hides all model edges that are covered by surfaces. In addition you can specify if hidden lines should be represented by *solid* or *dotted* lines.



You can rotate surfaced objects in any view mode using the *Trackball*. ***Draft-Board*** will automatically recalculate and redisplay the image in the new view orientation. Rotating an object in the Visible Lines (analytical view mode is slower than in all other modes.

In contrast to the view modes **Shaded** and **Visible edges** that are supported by ***OpenGL*** and represent all curves as pixel lines, the *analytical* view mode represents and calculates all model edges as ***NURB**-Splines*.

This view mode is slower than the ***OpenGL*** supported view modes **Shaded** and **Visible Lines**, but calculates precisely all edges including all sorts of intersection and hidden lines. This view mode is always recommended if 3D Models should be projected and dimensioned.

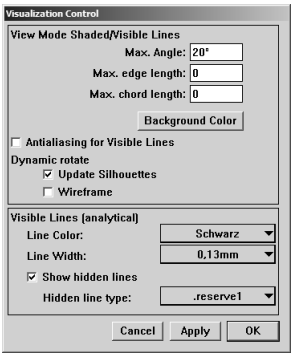
Preferences

The **Visualization** command in the **Preferences** submenu of the **Layout** menu allows you to set various options for the representation of surfaced objects.

Visualization

The **Visualization** command allows you to set some *Shaded Options*, to change the *Background color*, to select the *Appearance* of the removed *Hidden Lines* and to deactivate the *Default Lighting*.

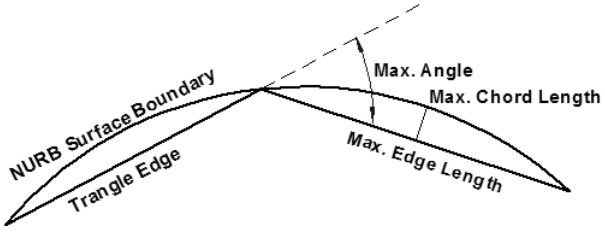
The **Visualization** command displays the following dialog box:



The **Visualization** dialog box contains the following elements:

Shaded/Visible Lines

For the view modes **Shaded** and **Visible Lines** all ***NURB** Surfaces* are triangulized, what means, that all non-planar ***NURB** Surfaces* are approximated by small triangles. The settings **Max. Angle**, **Max. Edge Length** and **Max. Chord Length** describe the ratio of the ***NURB**-Surface Edge* to the adjacent *triangle edge*.



Max. Angle This value describes the maximum possible *Exterior Angle* between two approximation triangles. Allowed are values **> 0°**. The default

value is **20°**. The smaller the value the larger becomes the number of facets. This increases the precision of the model representation, but also increases the calculation time.

Max. Edge Length This value describes the maximum possible *Edge Length* of a approximation triangle. A small value increases the precision of the model representation, but also increases the calculation time. When **0°** is entered, this parameter is ignored. The default value is **0°**.

Max. Chord Length This value describes the maximum *Chord Length* between the ***NURB*** *Surface boundary* and the adjacent triangle edge of a approximation triangle. A small value increases the precision of the model representation, but also increases the calculation time. When **0°** is entered, this parameter is ignored. The default value is **0°**.

Background Color This button allows you to set the *Background Color* for the active view (*Sheet View* or *Detail View*). When you click this button the *Windows* or *Macintosh* ***Color Palette*** is displayed where you can specify a new background color. The selected background color applies only for the view modes **Shaded** and **Visible Lines**.

Antialiasing When you activate this option all edges of a model are *smoothed*, while the calculation time slows down. This option applies only for the view mode **Visible lines** not for the **Shaded** view mode.

Dynamic Rotation This option specifies, if a model when rotated by the **Trackball** will be represented in the view mode **Wireframe** or **Visible Lines** and if the model representation should be updated when the mouse button is released.

Visible Lines (analytical)

For the view mode **Visible Lines (analytical)** all model edges are recalculated as ***NURB*** *Splines*. This precise vectorized model representation takes longer than the pixel representation of visible edges. For the representation of the calculated ***NURB*** *Splines* the following setting are possible.

Visible Lines (analytical)

Line Color In this ***Pop-up*** *Menu* you can specify a line color for all *visible* edges.

Line Width In this ***Pop-up*** *Menu* you can specify a line width for all *visible* edges.

Show Hidden Lines

Hidden Line Type In this ***Pop-up*** *Menu* you can specify a line type for all *visible* edges.

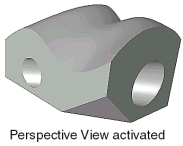
Perspective View

In ***DraftBoard*** surfaced models can be displayed in a *perspective view* using the **Perspective on/off** command in the **Views** menu.

The settings for this perspective view can be modified with the **Perspective Edit** command in the ***Views*** menu.

Perspective on/off

This command in the **Views** menu switches the perspective *on* or *off*. A *checkmark* will appear to the left of the name when the **Perspective** is activated.



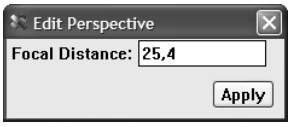
Perspective View activated

The **Perspective on/off** command is a toggle switch that can be used only with the view modes **Shaded** and **Visible Lines**. Perspective viewing is not available in the **Wireframe** view mode.

The default settings of the perspective can be modified with the **Perspective Edit** command in the **Views** menu.

Perspective Edit

The **Perspective Edit** command displays the following dialog box:



The **Perspective Edit** dialog window allows the following setting:

Focal Distance — This value determines the *Focal Distance* like the lens settings of a camera. The smaller the value the greater the perspective effect. The lowest allowed value is **0.1**. The default value is **25**.

Setting a large Perspective

Create for that example a simple cube and surface it using the **AutoSurface** command. Choose for the cube a **trimetric** view with a **shaded** view mode.

1. Choose the **Perspective on/off** command in the **Views** menu.
   The model is displayed in a perspective view according to the set default values.
2. Select the **Perspective Edit** command in the **Views** menu.
   The **Perspective Edit** dialog box is displayed.
3. Substitute the default value **25** for the **Focal Distance** by the value **5**.
   The model is displayed in a large perspective angle but smaller due to zoom factor used.
4. Enlarge the model using the **Zoom In** tool from the tool palette until the model fills the screen.

Setting a small Perspective

Create for that example a simple cube and surface it using the **AutoSurface** command. Choose for the cube a **trimetric** view with a **Shaded** view mode.

1. Choose the **Perspective on/off** command in the **Views** menu.
   The model is displayed in a perspective view according to the set default values.
2. Select the **Perspective Edit** command in the **Views** menu.
   The **Perspective Edit** dialog box is displayed.
3. Substitute the default value **25** for the **Focal Distance** by the value **200**.
   The model is displayed in a smaller perspective angle but larger due to zoom factor used.
4. Reduce the model using the **Zoom Out** tool from the Tool palette until the model fills the screen.

Illumination

For the visualization of surfaced objects you can either use the *default light setting* or specify and place you own light sources.

Default Lighting — The *Default Lighting* consists of a single *Ambient Light* and two *Parallel Lights* and is set automatically for all new drawings. It cannot be modified since the corresponding light sources are not placed as objects in the drawing, but it can be deactivated using the **Visualization** command from the **Preferences** submenu in the **Layout** menu.

Besides the *Default Lighting* ***DraftBoard*** offers three types of light sources for setting the illumination.

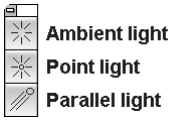
Ambient Light	This light source provides a uniform illumination on all sides of your geometry.
Point Light	This light source provides a more directed illumination.
Parallel Light	This light source allows in contrast to the other two light sources to specify the *location* and the *direction*. The Parallel light source provides a linear type of illumination, similar to what a fluorescent tube might provide. It is the most directed source for illumination in ***DraftBoard***.

The *distance* between the illuminated object and the *parallel light* source is unimportant. Crucial is only the *direction* of the light source.

These light sources are normal ***DraftBoard*** objects, that can be moved, copied or deleted like any other object in DraftBoard. All light source attributes can be edited using the **Edit Objects** command. They also can be placed or relocated using the *Drawing Assistant*.

Show Light Palette

The *Light Palette* contains the tools to create *Ambient*, *Point*, and *Parallel* light sources.



The command **Show Light Palette** in the **3D** menu displays the movable light palette in the upper left corner of the ***DraftBoard*** *window*.

Before you place any light source you should construct a surfaced object to control the effect of the lighting.

Ambient Light

With the **Ambient Light** tool you place a light source providing a uniform illumination on all sides of your object. As a diffuse light source it affects only the brightness. You position the light source with a mouse click on the drawing area. Light sources can be placed in any plane like any other object.

In the *Status Line* you can enter a factor for the *Size* of the light source icon. The default factor is **1**. This factor does not effect the intensity of the light source or any other aspect of the source. *Color* and *Intensity* of the light source can be specified with the **Edit Objects** command in the **Edit** menu. The default color is *white* and the default factor for the *intensity* is **0.5**.

Placing an Ambient Light

1. Display the *Trackball* and choose an *Isometric* or *Trimetric view*.
2. Turn off the *Default Lighting* using the **Visualization** command from the **Preferences** menu in the **Layout** menu.
3. Select **Wireframe** in the **View Mode** submenu, since the light source symbol is only displayed in the wireframe view mode.
4. Select the **Ambient Light** tool from the ***Light*** *Palette*.

   The *Status Line* displays an entry box with the default factor **1** for the symbol **Size** of the light source.
5. Specify with a mouse click on the drawing area the *Location* for the *Ambient* light.

   An icon for the *Ambient Light* source appears on the screen in that location.

6. If the size of the light source is too small or too large, change the scale factor in the status line accordingly and press the **Enter** key.
7. Choose **Shaded** in the **View Mode** submenu to display your geometry with your specified light settings.

8. If necessary change the *color* and *intensity* of the light source using the **Edit Objects** command in the **Edit** menu (see as well *Editing Light sources* at the end of this chapter).

Point Light

With the **Point Light** tool you place a light source providing a more directed of your object.

You position the light source with a mouse click on the drawing area. Light sources can be placed in any plane like any other object.

The *Status Line* lists the ***x***, ***y***, and ***z*** *coordinates* for the exact *position* and the *factor* for the icon *size* of the light source. The default factor is **25,4**.

Color and *Intensity* of the light source can be specified with the **Edit Objects** command in the **Edit** menu. The default *color* is *white* and the default factor for the *intensity* is **0.5**.

Placing a Point Light

1. Display the *Trackball* and choose an *Isometric* or *Trimetric view*.
2. Turn off the *Default Lighting* using the **Visualization** command from the **Preferences** menu in the **Layout** menu.
3. Select **Wireframe** in the **View Mode** submenu, since the light source icon is only displayed in the wireframe view mode.
4. Select the **Point Light** tool from the ***Light*** *Palette*.

   The *Status Line* displays entry boxes for the ***x***, ***y***, and ***z*** *coordinates* for the exact *position* and an entry field for the *factor* for the icon *size* of the light source. The default factor is **25,4**.
5. Specify with a mouse click on the drawing area the *Location* for the *Point Light*.

   An icon for the *Point Light* source appears on the screen in that location.

6. If the size of the light source is too small or too large, change the scale factor in the status line accordingly and press the **Enter** key.
7. Choose **Shaded** in the **View Mode** submenu to display your geometry with your specified light settings.
8. If necessary change the *color* and *intensity* of the light source using the **Edit Objects** command in the **Edit** menu (see as well *Editing Light sources* at the end of this chapter).

Parallel Light

With the **Parallel Light** tool you place a light source providing a linear type of illumination, similar to the light of a fluorescent tube.

You position the light source by ***dragging*** a vector on the drawing area to specify the *location* and *direction* of the light source.

You can also click two points for the *startpoint* and the *endpoint* of the vector.

In the *Status Line* you can enter the ***x***, ***y***, and ***z*** *coordinates* for *Startpoint* and the *Endpoint* of the vector and a *factor* for the icon *size* of the light source. The default factor is **25,4**.

Color and *Intensity* of the light source can be specified with the **Edit Objects** command in the **Edit** menu. The default *color* is *white* and the default factor for the *intensity* is **0.5**.

Placing a Parallel Light

1. Display the *Trackball* and choose an *Isometric* or *Trimetric view*.
2. Turn off the *Default Lighting* using the **Visualization** command from the **Preferences** menu in the **Layout** menu.
3. Select **Wireframe** in the **View Mode** submenu, since the light source icon is only displayed in the wireframe view mode.

The *distance* between the illuminated object and the *parallel light* source is unimportant. Crucial is only the *direction* of the light source.

4. Select the **Parallel Light** tool from the ***Light*** *Palette.*

   The *Status Line* displays entry boxes for the ***x***, ***y***, and ***z*** *coordinates* for the *Startpoint* and the *Endpoint* of the vector and a *factor* for the icon *size* of the light source. The default factor is **25,4**.

5. ***Drag*** a vector on the drawing area for the *location* and the *direction* of the *Parallel light* source.

   An icon for the *Parallel Light* source appears on the screen with arrows aiming in the specified direction.

6. If the size of the light source is too small or too large, change the scale factor in the status line accordingly and press the **Enter** key.
7. Choose **Shaded** in the **View Mode** submenu to display your geometry with your specified light settings.
8. If necessary change the *color* and *intensity* of the light source using the **Edit Objects** command in the **Edit** menu (see as well *Editing Light sources* at the end of this chapter).

Modifying the Lighting

As you design your geometry and shade it, you may also want to view it under various lighting conditions, either by changing the layout or editing the light source.

If you want to see the effect of your lighting changes without having to continuously switch from the **Wireframe** to the **Shaded** view mode, create a ***Detail View*** *window* of your geometry. Your sheet view automatically becomes the **Top** view. You now can *move*, *add* or *delete* light sources from the **Top** view and see the effect on your shaded geometry in the **Detail view**. When you make changes this way, ***DraftBoard*** will automatically reshade your geometry.

Editing Light sources

1. Select the light source symbol in the **Wireframe** view mode.
2. Choose the **Edit Objects** in the **Edit** menu.

   The **Edit Objects** dialog box is displayed listing all attributes of the selected light source.
3. Modify the *Light Type*, the *Position*, the *Color,* the *Intensity* and the *Direction* of the selected light source, if a *Parallel light* source is selected.

   For the die *Light Color* you can specify in the entry fields *Red*, *Green* and *Blue* values between **0** and **1** (values greater **1** have no effect).

   For the *Light Intensity* values between **0** and **4** are recommended.
4. Select the **Shaded** view mode from the **View Mode** submenu to control the light settings.

Deleting Light Sources

1. Select the light source symbol in the **Wireframe** view mode.
2. Choose the **Delete** command from the **Edit** menu or press the **Backspace** key (*Macintosh:* ***Delete*** *key*).

   The light source is deleted.

Moving Light Sources

1. Select the light source symbol in the **Wireframe** view mode.
2. ***Drag*** the light source to the new location.

   When you want to move a *Parallel Light*, you should display first the *startpoint* and the *endpoint* of the *light vector* using the **Show Points** command in the **Layout** menu. For moving the light source you select both points, for changing the *position* you select only the *startpoint* of the vector and for changing the *direction* you select only the *endpoint* of the vector.

Exporting Surfaces

For the export surfaced objects ***DraftBoard*** offers several data interfaces:

More information about ***Import*** and ***Export*** *data interfaces* you find in *Chapter* ***11*** and in the *Appendix* of this manual.

NURB-Surfaces

IGES Use this data format when you want to export the calculated surfaces as ***NURB*** *Surfaces*, that you want to use in other ***CAD*** *programs*.

Polygon Surfaces

The following formats will tessallate 3D Models by creating triangulated surface patches. The export with these formats is based on the settings in the 3D Export dialog in the Preference submenu of the Layout menu.

DXF/DWG When you use the ***DXF/DWG*** *Export format* all calculated ***NURB*** *surfaces* will be converted to ***Polygon*** *surfaces*.

This format is recommended for exporting surfaces into *Animation* and *Render programs*.

STL This format exports only three-dimensional structured surfaces and surfaced objects as polygon meshes.

This format is recommended for prototyping.

SLP This format from **Pro/E** is recommended for Rendering 3D models.

Graphics

EPS For the export of surfaced objects (especially when using the **Shaded** and **Hidden Lines** view modes) into ***Text*** and ***Layout*** *programs* you should use the ***EPS*** *option*.

This format guarantees a high quality and allows to scale the graphic after the export.

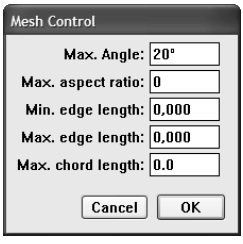
3D-Export Settings

All export settings will be configured automatically or will be defined using option buttons contained in the export dialogs for the specific data interfaces.

Only for the export of 3D models you can define some general defaults using the **Export 3D** command in the **Preference** submenu of the **Layout** menu.

Export 3D

When you select this command the following dialog box is displayed:



All settings in this dialog window apply only for the *Export* of 3D Models, while identical settings in the **Visualization** dialog box apply for the *representation* of 3D Models.

The following settings in the **Export 3D** dialog box are valid for all Export data interfaces, that tessellate 3D Models for the export (**DXF/DWG**, **STL**, **SLP**) and apply only for triangulated surfaces.

Max. Angle This value describes the maximum possible *Exterior Angle* between two approximation triangles. Allowed are values **> 0°**. The default value is **20°**. The smaller the value the larger becomes the number of facets. This increases the precision of the model representation, but also increases the calculation time.

Max. Aspect Ratio This value describes the aspect ratio of the *min. edge length* to *the max edge length* of a approximation triangle. The value **1** equals an equilateral triangle. When **0°** is entered, this parameter is ignored. The default value is **0°**.

Min. Edge Length This value describes the minimum possible *Edge Length* of a approximation triangle. A small value increases the precision of the model representation, but also increases the calculation time. When **0°** is entered, this parameter is ignored. The default value is **0°**.

Max. Edge Length This value describes the maximum possible *Edge Length* of a approximation triangle. A small value increases the precision of the model representation, but also increases the calculation time. When **0°** is entered, this parameter is ignored. The default value is **0°**.

Max. Chord Length This value describes the maximum *Chord Length* between the ***NURB*** *Surface boundary* and the adjacent triangle edge of a approximation triangle. A small value increases the precision of the model representation, but also increases the calculation time. When **0°** is entered, this parameter is ignored. The default value is **0°**.

Appendix

Operators and Units

This appendix describes mathematical and conditional operators.

Mathematical Operators

Many entry boxes accept mathematic, trigonometric, and exponential operators in the form:

funcName (arg1; arg2; . . . ; argN)

Operator	Example	
Addition	3+0,5	= 3,5
Subtraktion	3-0,5	= 2,5
Multiplikation	3*0,5	= 1,5
Division	3/0,5	= 6
Square Root	sqrt(2)	= 1,4142...
Parenthetical	3/(4+(1/0,5))	= 0,5
Scientific Notation	4e-3	= 0,004
Exponentiation	exp(2;7)	= 128
Sine of Angle	sin(30)	= 0,5
Cosine of Angle	cos(60)	= 0,5
Arctangent	atan(1;1)	= 45 (°)
Log (Base 10)	log(100)	= 2
Natural Log (Base e)	ln(2)	= 0,6931
Remove Fractional Part	truncate(1,4142)	= 1
Absolute Value	abs(-2)	= 2
Smallest Larger Integer	ceiling(1,4142)	= 2
Largest Smaller Integer	floor(1,4142)	= 1
Negative Value	neg(2)	= -2
Round	round(1,41)	= 1
Random Number	rand()	= (random number)

For divisions also a colon may be used instead of a slash.

Math operators are case-sensitive; so be sure to type them in lower case when you use them. You can use these operators to edit objects, make entries in the status line, and specify values in dialog boxes.

A

Conditional Expressions

Conditional expressions are particularly useful with *Parametrics* when an expression depends on a condition. For example, a flange might depend on the diameter of a pipe but should never be smaller than some minimum size. You can use two function forms of conditional expressions, **ifelse** and **RNG** (range).

ifelse (condition; expression1; expression2)

Condition	**The result of a boolean expression:**
	= = equals
	!= not equals
	> greater than
	>=greater than or equals
	< less than
	<=less than or equals
	&& logical **and**
	|| logical **or**
	! logical **not**
expression1	The value of the **ifelse** function if the condition is true.
expression2	The value of the **ifelse** function if the condition is false.

The following examples show boolean expressions and an **ifelse** statement:

Length > 5	The length is **greater 5**.
(L<3) || (W>=2)	Either the length is **less than 3** or the width is **greater than or equal to 2**.
3*ifelse ((A>B); A; B)	Evaluate three times the maximum of the variables **A** and **B**, returning **3*A** if **A** is larger than **B** and **3*B** if **B** is larger than **A**.

The range function is a more general form of conditional expression.

RNG (T; V1; N1; V2; N2; V3; N3; ... Vi; Ni; V, +1)

In this function, the first argument (**T**) is tested against every **N** value in the other arguments and returns the value (**V**) that lies between the two **N** values that bracket **T**. Each of the test **N** values must be greater than the preceeding **N**.

For example, the expression

RNG (A; -1; 10; 0; 20; 1)	returns	**-1**	if	**A < 10**
		0	if	**10 <= A<20**
		1	if	**A >= 20**

Units of Measure

You can use different units of measure such as inches (**"**), feet (**'**), inches and feet (**x'y"**), millimeters (**mm**), centimeters (**cm**), and meters (**m**). You can also mix the units in the mathematical expression as long as you label the units properly—for example, **10" + 25.4 cm**.

Nanoseconds

DraftBoard is well suited for computer hardware design, such as - diagrams of timing devices. For example, you can draw in nanoseconds (a billionth of a second).

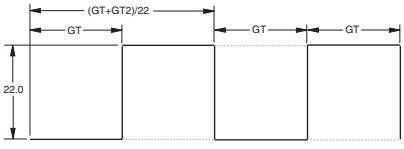
Designs which include nanoseconds typically consist of wave forms made of horizontal and vertical lines which mimic the functions of a timing device.

1. Specify **Millimeters** as the unit of measure in the **Units** submenu of the **Preferences** command in the **Layout** menu.
2. Choose **Grid** from the **Preferences** submenu.

3. Set the grid spacing to **34** and the number of subdivisions to **2**.
   A megahertz is approximately **34** nanoseconds and this becomes the base cycle, with **two** tickmarks for drawing half cycles.
4. Turn on the **Grid**.
5. Use the **Connected Line** tool to draw the wave form of the desired specifications.
6. Use the **Stretch** tool to stretch the wave form to half and full cycles.

Using parametrics for wave forms.

You can use parametrics to speed up this process, as illustrated below. The dotted lines are *constraint lines*.



You can place the *constraint lines* on another layer and hide them if you want.

Decimal Indicators

DraftBoard can handle both *American* and *European* decimal entries in the status line (**24,5** or **24.5**).

A

Special Characters

You can use special characters and accents which are available with your computer. Usually these characters are described in the user manual of your *Operating System*.

WIN

Special Characters for Microsoft Windows

You can enter many *special characters* from the keyboard. If necessary, unlock the keypad with the **Num Lock** key, and then hold down the **Alt** key and enter the numeric code for the character you want.

The character appears in the current font (including ***DraftBoard's*** Plotter font). The codes are listed in the *Microsoft Windows User's Guide*.

You can as well use the *Windows* ***Charset*** in the **Accessories** submenu of the **Start menu** to insert special characters into ***DraftBoard***.

The **ANSI**-Code you enter must have **4** digits. If the code exists only of **3** digits you must place a leading **0**, since the *four digit* code **0216** doesn't have the same result as **216**.

Examples:	°	=	**Alt 0176**
	Ø	=	**Alt 0216**
	ø	=	**Alt 0248**
	±	=	**Alt 0177**
	©	=	**Alt 0169**
	@	=	**Alt 64**
	€	=	**Alt 0128**

MAC

Special Characters for Macintosh-Computer

Many symbols and characters are available from the keyboard. To see the characters associated with a particular font, choose **Key Caps** from the **Apple** menu and select the font from the **Font** Menu. When you press the **Command**, **Option**, **Shift**, or **Control** key or combinations of these keys, the keyboard graphic changes to show which characters are available.

In addition of those characters you see in **Key Caps**, you can use the **Option** key to generate international accents in the current font, including ***DraftBoard's*** *Plotter font.*

Accent	*Key Combination*
é	**Option/e** then **letter** to be accented
è	**Option/~** then **letter** to be accented
ñ	**Option/n** then **letter** to be accented
ô	**Option/t** then **letter** to be accented
ü	**Option/u** then **letter** to be accented

The following list includes common symbols which are available in all fonts:

Accent	*Key Combination*
°	**Option/Shift 8**
ø	**Option O** (letter or number)
Ø	**Option/Shift O**
±	**Option/Shift =**

A

Data Interfaces

This appendix contains additional information on the data interfaces already described in the manual.

DWG/DXF Interfaces

In general, it is recommended that you show all your layers before you export from ***DraftBoard*** or any other CAD package so that you know what objects are exporting.

- ***DraftBoard*** will always export only the current model. If your file contains *Detail Views*, flatten them with the **Flatten View** command found in the **Views** menu. If you have dimensions already in the detail view and the view scale is set to **1**, you can group the dimensions and geometry together before flattening the view. This will bring the dimensions across to the current model for Export.
- ***AutoCAD*** doesn't accept any **&** character in a layer name. Rename therefore all layers having a **&** character in their name. Otherwise AutoCAD will not import the file and generate an error message instead.
- In case line patterns are disorted (too big or too small) in ***AutoCAD*** after importing a ***DraftBoard DXF*** *File*, change the ***AutoCAD*** variable **LTSCALE** to a smaller number to display the line patterns at an appropriate scale.
- In case that after importing a ***DraftBoard DXF*** *File* into **AutoCAD** the arrowheads of some angular dimensions are displayed in ***AutoCAD*** in the wrong direction (pointing towards the text rather than towards the extension lines), execute the **DIM** command in ***AutoCAD***, select the affected dimensions, and use the **UPDATE** command to display the arrowheads.

DXF Configuration Files

The two configurations files ***DXF****IN*.***CFG*** (Import) and ***DXF****OUT*.***CFG*** (Export) allow you to set additional parameters for *colors*, *line patterns*, *line weights* and *fonts*. These two configurations files work like filters considering all parameters specified in these files when importing or exporting ***DXF****- and* ***DWG****-Files*. Both configuration files must be in the **DraftBoard** folder. Before modifying any of these files you should read the related *readme files* ***DXF****IN*.***TXT*** and ***DXF****OUT*.***TXT*** for *Windows* and ***DXF*** *Doc* and ***DXF****-Info* for the *Macintosh*. Both files are ***ASCII*** *Files* that can be modified using any simple *Text Editor*.

How exporting affects some objects

It is difficult to exchange data between different graphics programs because each program defines geometric objects differently. Even though the **DXF/DWG** and **IGES** formats standardize this exchange, some objects do not transfer exactly. The following table tells you what to expect when you export ***DraftBoard*** objects.

DraftBoard	DXF/DWG
Hatches	Hatches are exported as groups, except the option **Ungroup hatches** is active, then they are converted in normal lines.
Detail views	All detail view will be exported, as long as not more than two models are displayed in ***DraftBoard***. In addition the selected ***AutoCAD*** *Version* must support **Modelspace** and **Paperspace** (*Version* ***11*** or higher). If not all *detail views* must be flattened.
Ellipses	Ellipses are exported into ***AutoCAD*** *Version* ***13*** or higher. For all previous versions ellipses are approximated by line segments.

The *Sheet View* will be exported into the AutoCAD ***Paperspace*** and *all Detail Views* into the *AutoCAD* ***Modelspace***.

Fonts	Fonts that are defined in the ***DXF*** *Configuration files* are maintained.
GD&T	**GD&T** frames will be correctly represented by lines and text, but cannot be edited any more, since they will be exported as grouped block.
Surface Symbols	Surface symbols will be correctly represented by lines and text, but cannot be edited any more, since they will be exported as grouped block.
Welding Symbols	Welding symbols will be correctly represented by lines and text, but cannot be edited any more, since they will be exported as grouped block.
Layer	Layer will be maintained.
Locked Objects	Objects are transferred correctly, but unlocked.
Sheets	Each sheet must be exported separately.
NURB-Splines	***NURB*** *Splines* are exported using ***AutoCAD*** *Version* ***13*** or higher. For all previous versions ***NURB****-Splines* are approximated by line segments.
NURB Surfaces	***NURB*** *Surfaces* are exported as **Meshes** (triangulized surfaces) using ***AutoCAD*** *Version* ***13*** or higher. All previous versions don't accept ***NURB*** *Surfaces*.
Smart Walls	Smart walls will be represented by simple lines.
Text	Multi-line text in a text frame is exported as separate text lines.
Tolerances	*Tolerances* are exported using ***AutoCAD*** *Version* ***13*** or higher. In all previous versions tolerances will go lost, but the dimensions are maintained.
Models	Since ***AutoCAD*** supports only one *Modelspace*, only one model can be exported from ***DraftBoard*** into ***AutoCAD***. If in ***DraftBoard*** 2 or more models are displayed in *detail views* on the current sheet, all views must be exported. This will be automatically checked when exporting ***DWG****/****DXF****-Files*.

DraftBoard	IGES
Crosshatching	The crosshatching is converted to symbol representation.
Detail views	The view is lost but the dimensions and text are retained.
Diameter dimension	The leader style may change.
Fonts	Fonts are not maintained.
GD&T symbol	The feature control frame is converted to a group of non-**GD&T** geometry.
Surface symbol	The **Surface** symbol is converted to symbol representation.
Welding symbol	The **Welding** symbol is converted to symbol representation.
Grouped geometry	The geometry is retained but it is no longer grouped.
Layers	The layers are lost but the geometry is retained.
Locked geometry	The geometry is retained but it is no longer locked.
Multiple sheets	Each sheet must be exported individually.
Radial dimensions	The leader style may change.
Smart walls	The double lines are converted to symbol representation.

If there are any problems with your exported ***IGES*** *files* you should use before creating any ***DXF*** or ***IGES*** *file* the **Remove Dots**, **Check Dimension** and **Select Duplicates** commands in the **Utility** menu.

In case these commands are not listed in the **Utility** menu, you must add them to this menu as it is described in the *Customization* section in *chapter* **4** of this manual.) In addition you should read the text files *SelCopy.****txt***, *AntiDot.****txt*** and *CheckDim.****txt*** in the **Function** folder in the **DraftBoard** directory.

Text	A multiple line note becomes many single lines.
NURBs	***NURB*** *Splines* are supported.

Plotter Fonts

A On the *Macintosh* only the font *Plotter* is available.

You can use with ***DraftBoard*** all ***TrueType*** and ***PostScript Fonts*** available on your computer. In addition ***DraftBoard*** offers own *Plotter Fonts* that are listed below:

The *Windows* version of ***DraftBoard*** contains in the **Fonts** subdirectory in the **DraftBoard** folder sample files that show for each character the related ***ANSI Code***.

In addition to the plotter fonts shipping with ***DraftBoard*** you can use all plotter fonts with the ***.shx*** *Format*. You just must copy these ***.shx*** files into the **Fonts** subdirectory of the **DraftBoard** folder. See as well in the *Text* chapter of this manual.

Plotter

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnoprstuvwxyz
ÄÖÜäöü^°!"§$%&/()=?'+'−*/#'_.,:;<>
1234567890

Plotter Extended

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnoprstuvwxyz
ÄÖÜäöü↑°!''§ $%&/()=?'+'−*/#'_.,:;<>
1234567890

Plotter Greek

ΑΒΧΔΕΦΓΗΙϑΚΛΜΝΟΠΘΡΣΤΥΩΞΨΖ
αβχδεφγηιϑκλμνοπρστυ∈ωξψζ
^°!"$%&/()=?'+'−*/#'_.,:;<>
1234567890

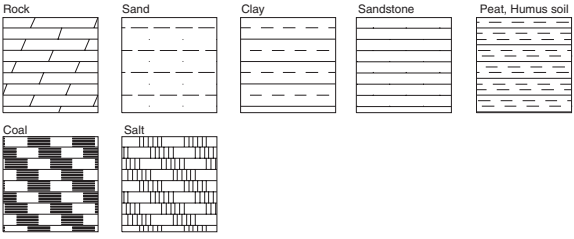
Plotter Roman

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnoprstuvwxyz
ÄÖÜäöü^°!"§$%&/()=?'+'−*/#'_.,:;<>
1234567890

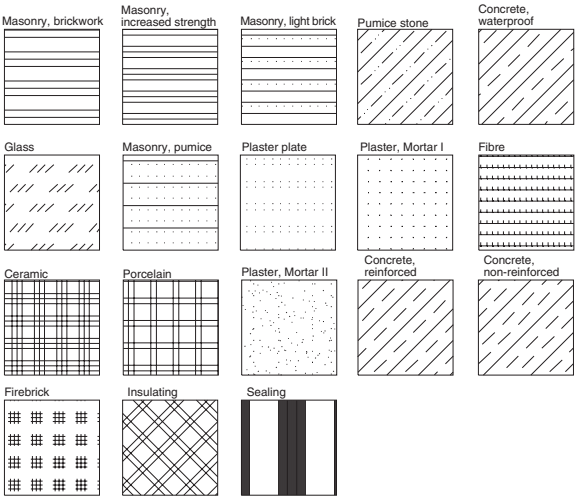
Crosshatch Patterns

DraftBoard offers a multitude of associative *DIN* and *ANSI hatch patterns*, of which you can modify the *spacing* and *angle*. You will find these hatches organized in groups such as *metals*, *fluids* etc. in the **Hatch** dialog box. All hatch patterns contained in this library are listed below.

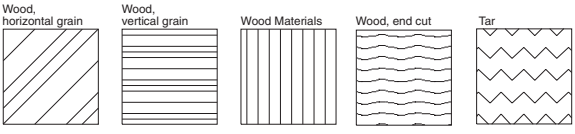
Mineral Materials



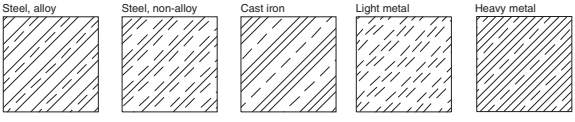
Other natural Materials



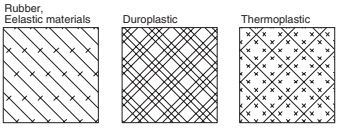
Natural Plant Materials



Metal

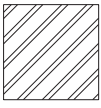


Synthetic Material



Fluids

Solids



A

DraftBoard.ini

The **DraftBoard.ini** file contains all default settings ***DraftBoard*** uses for any new drawing. The **DraftBoard.ini** file is an ***ASCII*** *text file* that can be opened and edited with any text editor.

To make changes to the **DraftBoard.ini** file open it from any text editor and make the desired changes. Save the file as a text file. The next time ***DraftBoard*** is launched it will take account for the new setting made in the **DraftBoard.ini** file.

The **DraftBoard.ini** file contains the following sections:

[User]

This section contains the *Name* of the registered user and the registered company

Name=
Company=

Both names will be automatically entered when installing ***DraftBoard***.

[Defaults]

In this section you can enter the *Path* and *Name* for the ***DraftBoard*** *Preferences file* **Prefs.vlm** as follows:

PrefFile=c:\Programs\DraftBoard\prefs.vlm

You only should specify a path for the *Preferences file* in case you don't want to use the *Preferences file* in the **DraftBoard** folder.

Creating a individual Preference files for a Network version

When you want to create an individual *Preference file* for each user of a ***DraftBoard*** *network version* you proceed as follows:

1. Create on the *Network Server* a folder (for example named **Preference**) for all *Preference files*.
2. Create folders for each user in this folder.
3. Copy the files **DraftBoard.ini** and **Prefs.vlm** in each folder.
4. Specify in each **DraftBoard.ini** file in the **Default** section the *Path* for the *Preference file* for the specific user:

   PrefFile=d:\Preferences\Smith\prefs.vlm

   where **Preferences** in this example is the name of the general *Preferences* folder and **Smith** the folder name of one user.
5. Select the ***DraftBoard*** *Program Icon* on the Windows Desktop.
6. Press the right mouse button and select **Properties** in the displayed menu.
7. Enter in the **Target** entry box behind **DraftBoard.exe** the path for the **DraftBoard.ini** file for the respective user as follows:

   C:\Programs\DraftBoard\DraftBoard.exe /
   i:d:\Preferences\Smith\DraftBoard.ini

;FILE LOCKING

FileLock=ON

When **File Locking** is activated (**= on**) all users can access a file in a network, but only the user who opened the file first has the right to save this file. For a single user license **File Locking** should be deactivated (**=off**).

File Locking works only if there exists a file named **public.key** on the *File Server* on the network. This Public.key file can be created with any text editor. The **Public.key** file can be even empty, it should be only placed in the main directory of the *File Server*.

;DISPLAYED FONTS

This section let you specify the number of fonts that will be displayed in the **Font** submenu from the **Text** menu. Fonts that do not show up in this menu can be accessed in the **Other** dialog box.

If you specify

MaxFonts=30

a maximum number of 30 fonts will be displayed.

;SCREEN

This option let you set the size of the screen that ***DraftBoard*** opens to.

Screen=FULL

opens to a maximized window.

Screen=Standard

opens to the standard size, which is almost maximized, but with room for icons at the bottom. If no value is specified, the screens opens to the standard size.

[RecentFiles]

The maximum number of files that can be displayed in the *Recent File List* at the end of the **File** menu is **8**.

Files=6

[Symbol]

This section contains the path of the **Symbol** folder relative to the **DraftBoard** folder

VStandard=Symbols

All further paths specified describe the absolute paths for the symbol palettes in case you added them to the **Utility** menu using the **Add command** in the **Utility** menu.

VMech=C:\Programs\DraftBoard\Symbols\Engineering
VArch=C:\Programs\DraftBoard\Symbols\Architecture

[SymTab]

This section contains the position of the **Parameter** dialog box that you can open in the **Symbols** dialog box.

LISTBOX=[80 140 356 358]

[Palettes]

This section contains all palettes available in ***DraftBoard***. Behind each palette name the position of the palette is specified and in a second section if the palette is visible or not (**true/False**):

Misc=190 80
Windows=100 275
Drawing=130 275
Bom=30 350
Dimension=10 345
Functions=30 440

MiscVisible=false
WindowsVisible=false
DrawingVisible=false
BomVisible=true
DimensionVisible=false
FunctionsVisible=false

Customer Services

Thank You for joining the world of ***DraftBoard***.

Please take your time and read the following lines carefully. They contain important informations concerning support and customer services..

Registration

By registering your new software, you'll receive:

- The right to *Updates* and *Upgrades*,
- Replacement of damaged media,
- Notification of product *updates*, *upgrade* and *new products*,

Filling out and returning the registration card is a painless process. Don't forget to enter the name of the respective user, even if the software is registered to a company.

Media Replacement

If your product is defective or damaged, all defective or damaged media will be replaced at no cost during the 3 month warranty period.

Updates and Upgrades

As registered user you will be notified of product *updates*, *upgrade* and *new products*,

- **Updates**: When we make minor improvements to a product, we will release an updated version.
- **Upgrades**: As we continually drive our technology forward, we may develop significant new features that enhance our existing offering. Such developments are incorporated into a new version of the software.

Product Support

All registered ***DraftBoard****-Customers* may call directly POSH GmbH for any support issues.

POSH GmbH
Burgweg 79
63897 Miltenberg

www.draftboard.de

Telefon: +49 (0) 93 71 - 65 297
Telefax: +49 (0) 93 71 - 69 241

E-Mail: support@draftboard.de

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